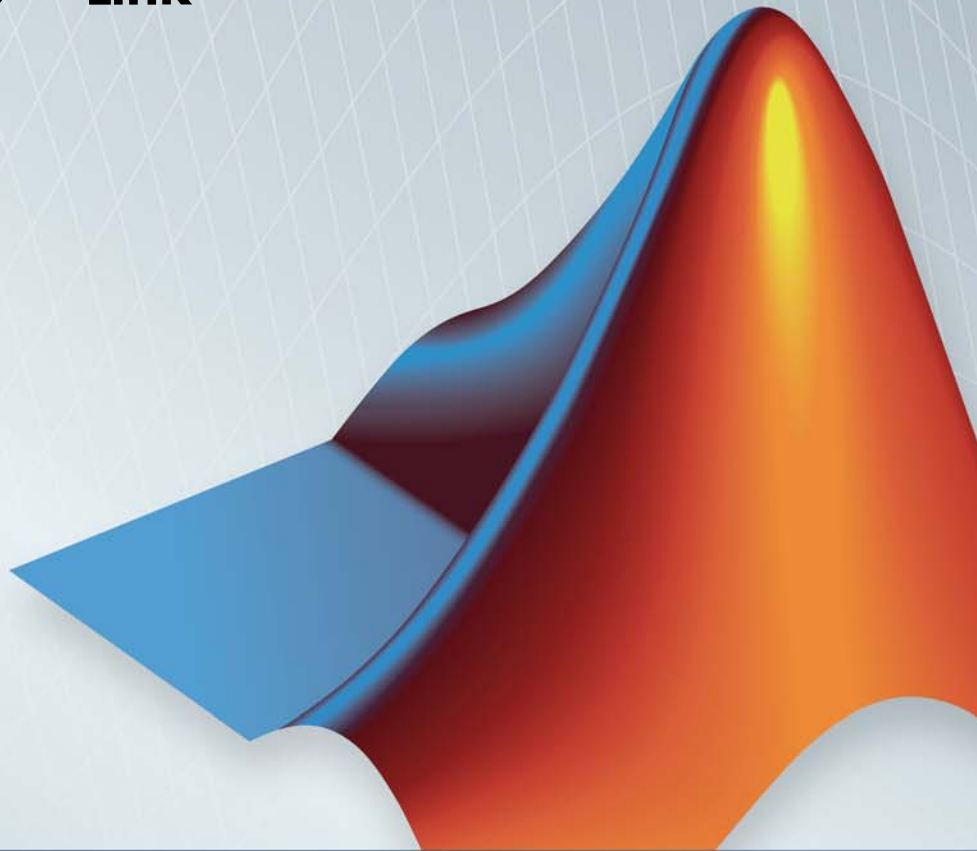


SimMechanics™ Link

User's Guide

R2013b



MATLAB®



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SimMechanics™ Link User's Guide

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Product Description

In this section...
“Product Definition” on page 1-2
“Using SimMechanics Link and SimMechanics Software Together for Complete CAD Translation” on page 1-2

Product Definition

Computer-aided design (CAD) is an integral part of engineering design in many industries. CAD tools allow engineers to model their mechanical systems in 3-D space. Although this approach is excellent for geometric modeling, incorporating controllers into this environment is difficult. Simulink® with SimMechanics software uses a block-diagram schematic approach for modeling control systems around mechanical devices. The SimMechanics Link utility bridges the gap between geometric modeling and block diagram modeling and simulation, by combining the power of Simulink and SimMechanics software with CAD.

Using SimMechanics Link and SimMechanics Software Together for Complete CAD Translation

Tip You can start learning how to use SimMechanics Link software with the “Watch a CAD Import Video” on page 1-12 section.

With the SimMechanics Link utility, you can create a SimMechanics model from a CAD assembly, in two steps.

Exporting CAD Assemblies Into Physical Modeling XML

The first translation step is to use the SimMechanics Link exporter to create an intermediate Physical Modeling XML file from a CAD assembly. Using SimMechanics software, you can then import that XML file to automatically generate a SimMechanics model.

With SimMechanics Link export, you initiate the translation of CAD assemblies into dynamical block diagram models. You export:

- CAD assemblies into Physical Modeling XML format. The XML file captures the mass and inertia of each part in the assembly and the constraint definitions between parts.
- Graphics files to define the body geometries of the assembly parts. The graphics files capture the body geometries of the assembly parts.

Importing Physical Modeling XML to Generate SimMechanics Models

The second translation step is to import the Physical Modeling XML to generate the SimMechanics model, then use that model together with the body geometry graphics files to simulate and visualize the original mechanical system.

- The XML representations of parts and constraints become bodies and joints in a SimMechanics model.
- The generated SimMechanics model uses the exported body geometry graphics files to visualize the bodies.

Creating a Custom Link using the SimMechanics Link API

For CAD platforms not directly supported, you can use the SimMechanics Link API to connect to your CAD platform API. You create a custom export link that achieves the same export results as for supported CAD platforms, export of the XML and body geometry files.

Related Products

In this section...
“Required Products” on page 1-4
“Other Related Products” on page 1-4

Required Products

SimMechanics Link software requires MATLAB® software.

Supported Operating Systems

You can use the SimMechanics Link product on any operating system that supports MATLAB.

Supported CAD Platforms

To build and export CAD assemblies, you need a CAD platform. Such a platform must either be supported directly by the SimMechanics Link utility or have an API you can use to write a custom SimMechanics Link interface.

Other Related Products

SimMechanics Product

Note The SimMechanics Link utility does not require the SimMechanics, Simscape™, or Simulink products.

The Physical Modeling XML and body geometry graphics files exported by the SimMechanics Link utility are intended for use with the SimMechanics product.

Physical Modeling Product Family

Use the Physical Modeling product family to model physical systems in Simulink. In addition to SimMechanics software, it includes:

- Simscape the platform and unifying environment for Physical Modeling products
- SimDriveline™ for modeling and simulating drivetrain systems
- SimElectronics® for modeling and simulating electronic systems
- SimHydraulics® for modeling and simulating hydromechanical systems
- SimPowerSystems™ for modeling and simulating electrical power systems

Install and Register SimMechanics Link Software

In this section...

“Installation Requirements” on page 1-6

“Download SimMechanics Link Software” on page 1-7

“Install SimMechanics Link Software” on page 1-7

“Register SimMechanics Link Utility with CAD Platform” on page 1-8

“Link External Application to SimMechanics Link Software” on page 1-8

“Register MATLAB as Automation Server” on page 1-8

“Unregister SimMechanics Link Software” on page 1-10

Installation Requirements

Before installing the SimMechanics Link utility, check that an active installation of the following software exists on your computer:

- MATLAB
- Supported CAD platform

MATLAB and SimMechanics Link must belong to the same release. For example, if your MATLAB release is R2012b, then your SimMechanics Link release must also be R2012b. Combining different release numbers can cause installation errors.

SimMechanics Link supports three CAD platforms:

- SolidWorks®
- Autodesk Inventor®
- PTC® Creo™ (Pro/ENGINEER®)

You can use the SimMechanics Link utility to export a CAD assembly from any of these CAD platforms. Note that MATLAB, SimMechanics Link, and your CAD platform must share the same architecture (e.g. 64-bit).

Download SimMechanics Link Software

You can download SimMechanics Link software directly from the MathWorks® website:

- 1 Visit the SimMechanics Link download website at http://www.mathworks.com/products/simmechanics/download_smlink.html.
- 2 Select the software version to install.
- 3 Click **Submit**.
- 4 Save the installation files in a convenient folder.

Do not extract the zip file.

Install SimMechanics Link Software

Install SimMechanics Link software from the MATLAB command line:

- 1 Start MATLAB.

Note You may need administrator privileges to complete the installation.

- 2 At the MATLAB command line enter:

```
path(path, '<installation_file_folder>')
```

replacing <installation_file_folder> with the path to the folder with the installation files.

- 3 At the MATLAB command line, enter:

```
install_addon('<zip_file_name>.zip')
```

replacing <zip_file_name> with the name of the zip file that you downloaded (e.g., smlink.r2012b.win64). The command extracts the zip archive files to the MATLAB root directory.

Register SimMechanics Link Utility with CAD Platform

Complete the installation by registering your the SimMechanics Link utility with your CAD platform. The registration procedure makes SimMechanics Link available in your CAD platform as an Add-In tool. Once you have completed the linking procedure, you can use the Add-In tool to export a CAD assembly directly from your CAD platform.

The registration procedure is different for each supported CAD platform. The following table provides platform-specific registration information. Click the link that matches your CAD platform, and complete the registration procedure.

To register with CAD platform...	...Click here
Autodesk Inventor	“Register SimMechanics Link with Inventor®”
PTC Creo (Pro/ENGINEER)	“Register SimMechanics Link with Creo”
SolidWorks	“Register SimMechanics Link with SolidWorks”

Link External Application to SimMechanics Link Software

You can link an unsupported CAD platform or other external application to SimMechanics software. For this task, SimMechanics Link provides an application programming interface (API) with a set of functions that you can use to create a C/C++ custom export module. For an overview of custom export using the API, see “Custom Export with SimMechanics Link API” on page 5-2.

Register MATLAB as Automation Server

Each time you use the SimMechanics Link utility with a CAD platform or other external application, the utility attempts to connect to MATLAB.

Registration Requirements

Successful connection requires the following to be true:

- Matching MATLAB and SimMechanics Link release numbers (e.g. both release numbers R2012b)
- MATLAB registration as automation server.

Enable Automation Server Mode

You can register MATLAB as an automation server in two ways:

Condition	Registration Procedure
MATLAB session open in desktop mode	At the MATLAB command line, enter <code>regmatlabserver</code> . The command registers the current MATLAB session as an automation server.
	At the MATLAB command line, enter <code>enableservice('AutomationServer',true)</code> . The command enables the current MATLAB session as an automation server.
MATLAB session not open	At the operating system command prompt, enter <code>matlab -automation -desktop</code> The prompt starts a new MATLAB session in automation server mode.
	At the operating system command prompt, enter command <code>matlab -regserver</code> . The command opens a new MATLAB session in automation server mode. You can close the MATLAB session.

A single MATLAB automation server registration can be active at a time. If multiple MATLAB sessions are open in your system, you must *first* disable

the active registration and *then* register the desired MATLAB session as an automation server using the `regmatlabserver` command.

Caution If your system does not have an active MATLAB automation server registration, SimMechanics Link issues a error when it attempts to connect. In the event of a connection error, check that a MATLAB automation server is active in your system. If necessary, register MATLAB as an automation server.

Connection from External Application to MATLAB Automation Server

Invoking the SimMechanics Link utility from an external application produces one of the following results:

Condition	Required Action	Result
No MATLAB session open	None	<ul style="list-style-type: none"> • New MATLAB session opens in automation server mode • SimMechanics Link connects to MATLAB automation server
MATLAB server open in automation server mode	None	<ul style="list-style-type: none"> • SimMechanics Link connects to MATLAB automation server
MATLAB session open in desktop mode	Register MATLAB session as automation server. See “Enable Automation Server Mode” on page 1-9.	<ul style="list-style-type: none"> • SimMechanics Link connects to MATLAB automation server

Unregister SimMechanics Link Software

SimMechanics Link contains no uninstaller. If you no longer wish to use the SimMechanics Link utility in your CAD platform, you can unregister the

utility. The following table provides information on the unlinking procedure for each CAD platform. Click the link that matches your CAD platform.

To link CAD platform...	...click here
Autodesk Inventor	“Register SimMechanics Link with Inventor”
PTC Creo (Pro/ENGINEER)	“Register SimMechanics Link with Creo”
SolidWorks	“Register SimMechanics Link with SolidWorks”

To register a different version of SimMechanics Link with your CAD platform, first unregister any currently registered version you may have. Then, register the desired version. To register and unregister the utility, follow the links provided in the previous table.

Watch a CAD Import Video

In this section...
“About the Example” on page 1-12
“Watch the Example Video” on page 1-12
“Learn More About the Example” on page 1-13

About the Example

The example consists of a series of steps that:

- Start with a CAD assembly.
- Show how, with SimMechanics Link and SimMechanics software, to translate the assembly into a SimMechanics model through export and import.
- Show how to modify the original assembly, then reexport and reimport it with successive changes that modify the imported model.
- Show how to manually modify the initial generated model, then update it with changes to the original assembly, without losing your manual changes to the model.

The assembly models a double pendulum, subsequently modified to a triple pendulum.

What the Example Requires

The complete procedure requires the original CAD assembly and platform, as well as SimMechanics Link and SimMechanics software.

Watch the Example Video

A video of the example is available at the MathWorks Web site, www.mathworks.com. If you are reading this in a browser and have access to the Internet, click here to access the example page.

What the Video Requires

The video requires a Web browser with a compatible streaming video application.

Learn More About the Example

The example video performs the steps of the case study, Updating and Retranslating a CAD Pendulum.

CAD Export

The SimMechanics Link exporter allows you to translate a machine defined externally (such as a computer-aided design assembly) into an intermediate representation. From this intermediate representation, you can generate a SimMechanics model representing the original machine and simulate its motion in the Simulink environment. The intermediate representation allows you to separate the export of external machine data with the SimMechanics Link utility and the generation of the dynamic model with the SimMechanics importer.

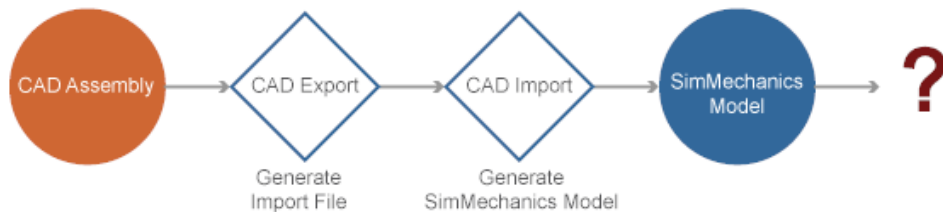
- “About CAD Translation” on page 2-2
- “Export Robot CAD Assembly” on page 2-6
- “Export Stewart Platform CAD Assembly” on page 2-8
- “Guidelines for CAD Translation” on page 2-10
- “Export and Re-Export CAD Assembly” on page 2-15
- “Retranslate CAD Assembly” on page 2-21
- “Troubleshoot CAD Export Issues” on page 2-27

About CAD Translation

In this section...
“Software Requirements” on page 2-2
“CAD Export” on page 2-3
“CAD Import” on page 2-5

CAD translation is a modeling process that converts a CAD assembly into a SimMechanics model. The complete process contains two sequential steps:

- 1** CAD Export — The SimMechanics Link utility generates an import file. The file reflects assembly structure and contains part parameters.
- 2** CAD Import — SimMechanics interprets the import file and generates a new model. The model structure and part parameters mirror the original CAD assembly.



Software Requirements

The complete CAD translation process requires the following software.

Software	Notes	CAD Export	CAD Import
CAD Platform		✓	
MATLAB	Registration as computing server required	✓	
SimMechanics			✓
SimMechanics Link		✓	

Note CAD Export does not require SimMechanics, Simscape, or Simulink products.

CAD Export

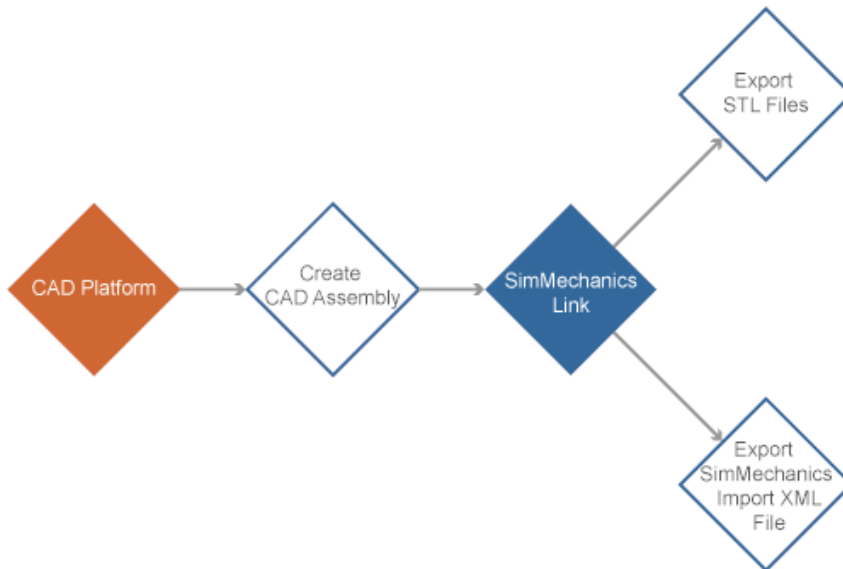
CAD export relies on the free utility SimMechanics Link. You can download the utility directly from the Mathworks website. See “Install and Register SimMechanics Link Software” on page 1-6. Following download and installation, registration of SimMechanics Link with a supported CAD platform adds the utility as an Add-In tool. You can now export a CAD assembly.

During CAD export, the SimMechanics Link utility generates one XML file and a set of STL files. The following table describes each file type.

File Type	Quantity	Purpose	Required for Model Generation	Required for Model Visualization
XML	1 total	Provide structure and parameters of CAD assembly in	✓	

File Type	Quantity	Purpose	Required for Model Generation	Required for Model Visualization
		SimMechanics format		
STL	1 per distinct CAD part	Provide 3-D surface geometry of CAD parts		✓

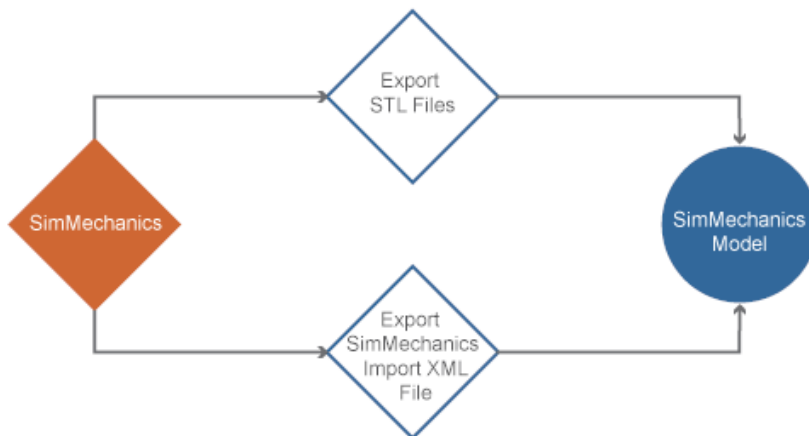
The files contain the assembly structure and part parameters required to generate an equivalent SimMechanics model. Assembly structure includes assembly-subassembly dependencies, which translate into SimMechanics system-subsystem dependencies. Part parameters include reference frames, mass and inertia, color, and location of part STL files.



CAD Export Process

CAD Import

CAD import is the second and final step of CAD translation. During CAD import, SimMechanics interprets the XML import file and automatically generates the corresponding model. SimMechanics imports CAD parts as rigid bodies, and CAD constraints as joints. The set of STL files provide the 3-D surface geometry of each SimMechanics rigid body.



CAD Import Process

CAD Import does not require access to the original CAD assembly or associated CAD platform.

Access to the surface-geometry STL files is not required for simulation, but it is required for visualization. You can simulate an imported model that contains no STL files, but the Mechanics Explorer visualization utility cannot display a representation of the model.

Export Robot CAD Assembly

In this section...
“CAD Files” on page 2-6
“Platform-Specific Steps” on page 2-6
“CAD Export Steps” on page 2-7

This example guides you through the export procedure for a CAD assembly with name `sm_robot.xml`. The assembly models an industrial robot arm that contains a grip subassembly.

CAD Files

Required CAD files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
matlabroot\toolbox\physmod\smlink\smlinkdemos\robot
```

where `matlabroot` is the directory of your MATLAB installation (for example, `c:\program files\MATLAB\R2012b`).

Platform-Specific Steps

The procedure in this example is general and applicable to all supported CAD platforms. For platform-specific steps, click the appropriate link in the following table.

For CAD platform...	...click here
Autodesk Inventor	“Export Robot Assembly from Autodesk Inventor Software” on page 4-8
PTC Creo	“Export Robot Assembly from Creo Software” on page 4-13
SolidWorks	“Export Robot Assembly from SolidWorks Software” on page 4-2

CAD Export Steps

Complete the following steps to export the robot CAD assembly in second-generation format. The procedure generates one SimMechanics Import XML file and a set of STL geometry files. Import the files into SimMechanics to generate a model that replicates the CAD assembly.

- 1** In a supported CAD platform, open assembly file `sm_robot`.
- 2** On the CAD menu bar, click **SimMechanics Link**.
- 3** Select **Export into SimMechanics First Generation** or **Export into SimMechanics Second Generation**.
- 4** In the **Save As** dialog box, enter the export file name and click **OK**.

Export Stewart Platform CAD Assembly

In this section...
“CAD Files” on page 2-8
“Example Requirements” on page 2-8
“Platform-Specific Steps” on page 2-9
“CAD Export Steps” on page 2-9

This example guides you through the export procedure for a CAD assembly with name `stewart_platform.xml`. The assembly models a Stewart platform commonly used in flight-simulator systems.

CAD Files

Required CAD files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
matlabroot\toolbox\physmod\smlink\smlinkdemos\...  
...<cad_platform>\stewart
```

String `matlabroot` is the directory of your matlab installation (for example, `c:\program files\MATLAB\R2012b`). String `<cad_platform>` is one of two CAD platforms that contain the Stewart Platform example:

- Creo (Pro/ENGINEER)
- SolidWorks

Example Requirements

To successfully complete the example, you must have:

- Supported CAD platform
- MATLAB installation, registered as an automation server. See “Register MATLAB as Automation Server” on page 1-8.
- SimMechanics Link installation, registered with the appropriate CAD platform.

To link to...	See...
Creo (Pro/ENGINEER)	“Register SimMechanics Link with Creo”
SolidWorks	“Register SimMechanics Link with SolidWorks”

Platform-Specific Steps

The procedure in this example is general and applicable to all supported CAD platforms. For platform-specific steps, click the appropriate link in the following table.

For CAD platform...	...click here
PTC Creo	“Export Stewart Platform from Creo Software” on page 4-26
SolidWorks	“Export Stewart Platform from SolidWorks Software” on page 4-20

CAD Export Steps

Complete the following steps to export the robot CAD assembly in second-generation format. The procedure generates one SimMechanics Import XML file and a set of STL geometry files. Import the files into SimMechanics to generate a model that replicates the CAD assembly.

- 1** In a supported CAD platform, open assembly file `stewart_platform`.
- 2** In the CAD menu bar, click **SimMechanics Link**.
- 3** Select **Export into SimMechanics First Generation** or **Export into SimMechanics Second Generation**.
- 4** In the **Save As** dialog box, enter the file name and click **OK**.

Guidelines for CAD Translation

In this section...

“How CAD Assemblies Are Translated into SimMechanics Models” on page 2-10

“Prepare CAD Assembly for Import into SimMechanics Model” on page 2-12

How CAD Assemblies Are Translated into SimMechanics Models

During CAD export, SimMechanics Link generates an XML file that maps CAD entities onto equivalent SimMechanics entities. The following table indicates the correspondence between the two types of entities.

CAD Entity	SimMechanics First Generation Equivalent
Part	Body
Constraints (Mates)	Joint
Reference coordinate system	Unconnected coordinate system ports
Assembly Reference	Fundamental Root: Root Ground — Weld – Root Part
Subassembly	Subsystem
Subassembly Reference	Root Part
Fixed Part	Root Body – Weld – Body

Origins, References, Roots, and Root Bodies

Every CAD assembly has a single *assembly origin* and one or more *assembly references* that do not move with respect to the origin. The positions and orientations of all parts refer directly or indirectly to this origin.

A *root body* is a zero-mass, zero-inertia body used in the generated SimMechanics model to represent one or more assembly references. A root body is always welded to ground, so that its zero mass and zero inertia do not affect the model’s dynamics. A root body is necessary to represent a fixed

anchor for part constraints in the original assembly. This body can carry multiple coordinate systems for this purpose, while the single Ground block in the generated model can carry only one.

Assembly Origin Mapped to SimMechanics World Origin

CAD translation maps the CAD assembly origin to the World coordinate system origin.

Subassemblies and Hierarchies

You can isolate a collection of CAD components (parts and their constraints) into a *subassembly*. CAD translation converts subassemblies into SimMechanics subsystems.

The main assembly is like the trunk of a tree, and its subassemblies are like the branches of the tree. Subassemblies can have subassemblies, and so on. This tree is the assembly's *hierarchy*. Each CAD subassembly has its own *subassembly origin* and *references*. A *fixed part* of a CAD subassembly is a part that is welded to a subassembly reference. It cannot move relative to the subassembly origin.

For an example of subassembly hierarchy, see “Export CAD Robot Arm” on page 3-40.

Mass Properties of Assembly Parts

The CAD assembly's parts need to have masses and inertia tensors. When you generate the SimMechanics model, this mass property information is used to specify the properties of the SimMechanics Body block corresponding to each assembly part.

Constraint Geometries

The constraints in your CAD assembly restrict how the assembly's parts can move with respect to each other. Without any constraints, a pair of CAD parts can move with six unrestricted degrees of freedom (DoFs) relative to one another. Constraints between pairs of parts reduce the six to fewer DoFs. SimMechanics joints express DoFs between bodies because SimMechanics bodies by themselves carry no DoFs. CAD constraints and SimMechanics joints are complements of one another.

Each joint is connected to two bodies, each at a body coordinate system (CS). The constraint geometry determines the joints into which CAD export translates the constraints and controls the position and orientation of the body CSs. Each of these body CSs has an origin and axis triad fixed relative to its body. CAD translation creates body CSs on the bodies, as necessary, for connecting joints.

Reference Coordinate Systems

Your CAD assembly can contain reference coordinate systems inserted on parts or elsewhere in the assembly. In general, these coordinate systems are not associated with constraints.

Export of such reference coordinate systems is optional. If these coordinate systems are translated into a SimMechanics model, they appear as coordinate systems on Body blocks, but not associated with Joint connections. When the model is initially imported, the reference coordinate systems are unconnected.

Prepare CAD Assembly for Import into SimMechanics Model

You need to specify enough information in your CAD assembly for the SimMechanics importer to construct a valid model from the XML file.

CAD Export Requires an Assembly

When you export from your CAD platform, you must export a complete assembly into XML, not just a part. If you have only one part, you must embed it in an assembly.

Simplify Assembly with Subassemblies

Use subassemblies to organize your assembly hierarchically. This simplifies your subsequent SimMechanics model by grouping blocks into corresponding subsystems. Follow these guidelines to ensure that your CAD assembly translates into a functioning SimMechanics model:

- You must have at least one fixed part inside each subassembly. (For more about fixed parts, see “Subassemblies and Hierarchies” on page 2-11.)

- Put as many welded components as you can inside rigid subassemblies or combine welded components into a single equivalent part.

If your assembly has a group of parts that do not move relative to one another, model them as a single part, eliminating unnecessary Body and Joint blocks from the generated SimMechanics model.

Specify Mass Properties of Assembly Parts

Your CAD platform might compute masses and inertia values from the mass density and geometry of the assembly parts. Otherwise, you must specify the mass and inertia tensor with respect to the part's center of gravity. The SimMechanics Link utility computes the center of gravity of each part automatically.

See “Export a CAD Rigid Body” on page 3-4 for an example.

Insert and Name Reference Coordinate Systems

Depending on your CAD platform and SimMechanics Link interface, you might be able to insert reference coordinate systems on assembly parts or elsewhere in your assembly. If you choose to export these coordinate systems, translation creates corresponding unattached Body coordinate systems in the generated SimMechanics model. You control the selective export of these reference coordinate systems by giving names with a distinctive prefix to these coordinate systems.

You can use reference coordinate systems in several ways.

- Prepare a CAD assembly for manual addition or replacement of Joints after import. These manually added Joints are independent of Joints automatically generated from assembly constraints.

Some CAD constraints are not supported for export. When you translate an assembly into a model, constraints that fail to translate into moving Joint blocks appear as rigid Welds. To represent your assembly correctly requires manual replacement of these Welds with moving Joints.

- Prepare a CAD assembly for manual addition of Constraints, Drivers, Actuators, and Sensors after import. To attach these blocks to Bodies, you need additional Body coordinate systems, apart from the Body coordinate

systems automatically generated to connect Joints representing assembly constraints.

You might find it easier to prepare for manually adding Joints, Constraints, Drivers, Actuators, and Sensors by setting up reference coordinate systems before export and attaching these Body coordinate systems to manually added blocks after import.

Specify Constraint Geometries

You must specify the constraint geometry in the CAD assembly consistently and in enough detail to reconstruct the assembly's DoFs as joints. The relationship between constraints in CAD and SimMechanics joints is not, in general, a simple mapping. Some SimMechanics joints have only one DoF, while others represent more than one DoF. CAD translation often combines multiple DoFs into one joint. Constraint specification details depend on the specific CAD platform.

For an example of configuring constraints, see “Export CAD Robot Arm” on page 3-40.

Avoid Redundant Constraints

Keep constraints simple and few enough to avoid creating unnecessary joints in your SimMechanics model.

For example, consider three parts, P1, P2, and P3, in an assembly. Suppose P1 and P2 are constrained so that there is no movement possible between them. When you attach P3, you could put one constraint between P3 and P1 and the other between P3 and P2. This leads to a redundant joint in the SimMechanics model, making it harder to understand and troubleshoot than if you created only one constraint. In this example, it is better to create a constraint just between P3 and P2, since P2 cannot move with respect to P1 anyway.

Export and Re-Export CAD Assembly

In this section...

“About CAD Export” on page 2-15

“Configure SimMechanics Link in Supported CAD Platforms” on page 2-16

“Generate Physical Modeling XML File” on page 2-17

“Re-Export Assembly After Changes” on page 2-18

“Export Robot Arm Assembly” on page 2-19

“Re-Export a Robot Arm Assembly After Changes” on page 2-20

About CAD Export

This section explains at a high level how to export CAD assemblies from CAD platforms supported by the SimMechanics Link utility.

Requirements for Export with a Supported CAD Platform

The SimMechanics Link Reference documentation presents platform-specific information on:

- Linking a supported CAD platform with the SimMechanics Link utility
- Finding, changing, and applying export settings
- Exporting assemblies in the Physical Modeling XML format through the CAD platform interface

Requirements for Export with an Unsupported CAD Platform

If the SimMechanics Link utility does not support your CAD platform, you can still export assemblies to Physical Modeling XML. In this case, you must construct a custom exporter that utilizes the SimMechanics Link application program interface (API).

Configure SimMechanics Link in Supported CAD Platforms

Once it is linked to the SimMechanics Link utility, each CAD platform's interface has a SimMechanics Link menu that allows you to access the export settings pane or dialog box. Before you export an assembly, you can check, change, and apply the export settings.

Open Settings Pane or Dialog Box

- 1** Open your CAD assembly.
- 2** From the CAD platform's SimMechanics Link menu, open the settings interface.

At any time, you can:

- Apply your settings.
- Cancel your settings. You lose whatever new settings you have entered.

Configure Export Tolerances

In the settings interface, you can configure one or more of the export tolerances. Geometric and numerical differences smaller than the tolerances are treated as zero.

- **Linear tolerance** specifies the smallest significant difference in length.
- **Angular tolerance** specifies the smallest significant difference in angle.
- **Relative roundoff** specifies the smallest significant numerical difference.

Configure Reference Coordinate Systems for Export

In the export coordinate systems interface, you can require the export of all, some, or none of the reference coordinate systems in your assembly. You control selective export by adding a distinctive prefix to the names of assembly coordinate systems that you want to export.

- You can export all the reference coordinate systems by choosing export and specifying no prefix.

- You can export some of the reference coordinate system by choosing export and specifying a prefix. Reference coordinate systems labeled by names with that prefix are exported.
- You can also choose not to export any reference coordinate systems.

For example, if your assembly has 11 reference coordinate systems (five with names beginning with A_, three with names beginning with B_, and three with names beginning with C_), you can choose to export:

- All 11 coordinate systems by specifying no prefix.
- The five coordinate systems with names prefixed by A_ by specifying the prefix A_.
- The three coordinate systems with names prefixed by B_ by specifying the prefix B_.
- The three coordinate systems with names prefixed by C_ by specifying the prefix C_.

Note Your CAD platform and SimMechanics Link interface might not support inserting and exporting reference coordinate systems. For details, consult the SimMechanics Link Reference Documentation.

Apply Export Settings

Apply your export settings by clicking the **OK** button in the settings pane or dialog box. The settings interface closes.

Generate Physical Modeling XML File

To complete export of the assembly to a Physical Modeling XML file:

- 1** If you changed the assembly or any subassemblies, you must rebuild the assembly and resave it in its native format before exporting it to XML.
- 2** Export the assembly through the CAD platform's SimMechanics Link interface.

The assembly is saved in a new form as an XML file. The exporter displays a dialog box when it is finished.

The default XML file name and folder are the same as those of the CAD assembly file. With the export interface, you can change the XML file name and folder if you want.

Automatic Export of STL Files

In addition to the XML file, the exporter creates a stereolithographic (STL) file for each part in the assembly that represents the part's body surface geometry.

- The STL files automatically receive names based on their respective part names in the assembly.
- The STL files are saved in the folder where the XML file is saved.

Re-Export Assembly After Changes

The results of a CAD export are changed if you:

- Change the assembly or any of its subassemblies or parts
- Change the export settings. See “Configure SimMechanics Link in Supported CAD Platforms” on page 2-16.

Re-exporting an assembly after such changes is, by default, identical to the initial export steps. See “Generate Physical Modeling XML File” on page 2-17. With no changes to the XML file save procedure, your re-export overwrites the existing XML and STL files with the same names.

Tip For later use and comparison, you might want to preserve the older XML and STL files.

- You can use the old folder and give the re-exported XML file a new name. The exporter still overwrites the old STL files.
 - You can use a new folder and avoid overwriting any old files.
-

Consequences of Re-Exporting Assemblies for Generated Models

While the re-export procedure is the same as, or only slightly different from, the initial export, the resulting new Physical Modeling XML file represents a revised CAD assembly, not a new assembly. To import this XML file with the SimMechanics importer requires choosing how to implement the revisions:

- Create a new SimMechanics model
- Update an existing generated SimMechanics model

You can make more detailed adjustments to the second choice as well, using the persistent properties that the XML file retains when you revise and re-export an existing assembly. See “Retranslate CAD Assembly” on page 2-21.

Export Robot Arm Assembly

In this example, you export a CAD assembly for the first time into Physical Modeling XML.

- 1** From the SimMechanics Link demos folder, open the robot arm assembly file, `robot.ASSEMBLYFILETYPE`, using a supported CAD platform linked to the SimMechanics Link utility.

The example CAD files are under `$matlabroot/toolbox/phymod/smlink/smlinkdemos/`, in the subfolder appropriate to your supported CAD platform. The file extensions of these files are specific to each CAD platform.

- 2** If you want, open the settings pane or dialog box from the SimMechanics Link menu in your CAD interface. (See “Configure SimMechanics Link in Supported CAD Platforms” on page 2-16.)

Make any adjustments you want to the export settings and apply the settings. The settings interface closes.

- 3** From the CAD platform’s SimMechanics Link menu, open the export interface.

- 4 Change to a different folder to export the XML file. Leave the file name as the default, `robot.xml`.
- 5 Start the export. The exporter begins converting and saving the XML file. When export is complete, a message appears notifying you of export completion.

The XML file is saved to the folder you chose. The exporter also automatically writes the stereolithographic (STL) body surface geometry files to the folder you choose.

Re-Export a Robot Arm Assembly After Changes

In this example, you re-export the robot arm CAD assembly after making some changes. You save the new version of the XML file under a new name, but overwrite the old stereolithographic (STL) files.

- These changes can include modifications of the assembly or subassembly parts.
- They can also include modifications to the export settings. (See “Configure SimMechanics Link in Supported CAD Platforms” on page 2-16.)

With the assembly open from your CAD platform interface:

- 1 Open the export interface from the SimMechanics Link menu. Remain in the same folder in which you saved the original exported files.
- 2 Change the exported file name to `robot1.xml`.
- 3 Start the export.

The new XML file, with the new name, is saved in the same folder as the original export. The exporter overwrites the old stereolithographic files with new ones, named with the same names.

Retranslate CAD Assembly

In this section...

“Associativity and Updating” on page 2-21

“Working with Associativity in Common Updating Situations” on page 2-24

Associativity and Updating

To update an existing generated SimMechanics model with changes to its original external definition (a CAD assembly, for example), the intermediate Physical Modeling XML file and the model itself must retain information about the identities of at least some of its components. This section explains this “identity memory” or *associativity*.

What Is Associativity?

Associativity is a key concept for understanding the relationship between CAD assemblies and SimMechanics models based on them, and the export and updating process that defines SimMechanics models from CAD assemblies.

Associativity Between CAD Assemblies and SimMechanics Models

Associativity is a persistent (session-independent) parallel relationship among certain components of a CAD assembly, Physical Modeling XML files exported from it, and SimMechanics models generated from the XML files.

This relationship preserves the identities and parallelisms of certain CAD components and the corresponding imported components of the SimMechanics model. The SimMechanics Link exporter defines these unique identities from the CAD assembly components and embeds them in the exported Physical Modeling XML file. SimMechanics models generated from the XML file in turn retain these identities.

You actualize associativity when you generate a SimMechanics model from a CAD assembly. Associativity is a mapping between parts, constraints, and subassemblies in a CAD assembly and the corresponding Body and Joint blocks, coordinate systems, and subsystems in the SimMechanics model generated from that CAD assembly. It uniquely captures the identities of

these CAD components, their corresponding blocks, and their topology (how they are connected to one another).

Associativity is not completely symmetric between the CAD and SimMechanics worlds, because the translation process moves in one direction only, from CAD assembly to generated SimMechanics model.

When and Why You Need Associativity

Associativity is required for updating a generated SimMechanics model when its originating CAD assembly has been changed.

How Associativity Is Implemented

When you use the SimMechanics Link exporter to create a Physical Modeling XML file from a CAD assembly, these components receive unique XML identifiers. When you use the SimMechanics importer to generate a SimMechanics model from the XML file, the identifiers are preserved in the corresponding SimMechanics model features.

Parallel Identities Between CAD Assembly and SimMechanics Model Components Captured by Associativity

CAD Assembly Components	Corresponding Imported SimMechanics Model Components
Parts and grounds	Body and Ground blocks
Constraints between parts (allowed motions)	Joint blocks
Constraints between parts (positions and orientations)	Paired coordinate systems attached to Joints
Reference coordinate systems	Body coordinate systems unattached after import
Subassembly hierarchy	Subsystem hierarchy

Changing Assemblies, Generated Models, and Their Associativity

The associativity of CAD assembly and generated SimMechanics model is open, modifiable, and extensible. As long as a generated SimMechanics model

retains at least one associated imported component, this model retains some associativity with its originating CAD assembly.

Preserving associativity. You preserve the original associativity if you do not remove or reconnect associated components in either the CAD assembly or the SimMechanics model.

Changing the properties of an associated component, without removing or reconnecting it, both uses and preserves associativity.

Extending associativity. You extend the original associativity if you add new, associable components to the CAD assembly, export the assembly, and update-import the generated SimMechanics model. The new components generated in the updated SimMechanics model are associated with the new components of the CAD assembly.

Modifying associativity. You modify the original associativity if you remove or reconnect one or more associated components in the SimMechanics *model*.

- The associativity of the removed or reconnected associated components is destroyed.
- The associativity of the other associated components, and of the SimMechanics model as a whole, remains intact.
- You recreate the original associativity of the removed or reconnected components in the SimMechanics model if you reimport the unchanged components from the CAD assembly.

Replacing associativity. You replace the original associativity if you remove or reconnect one or more associated components in the CAD *assembly*.

Once you export the CAD assembly and update-import the SimMechanics model, the associativity of the removed or reconnected components is destroyed. In this case, the component is either connected in a new way, with a new associativity, or it is removed altogether.

Working with Associativity in Common Updating Situations

The unique parallel identities created by associativity allow you to revise and expand CAD assemblies, then export the changed CAD assemblies and update existing SimMechanics models based on them. While you can also create entirely new SimMechanics models from the updated XML, associativity saves the effort invested in editing and testing by reusing existing SimMechanics models.

The following translation cases cover the basic possibilities. You can combine some of them into more complex, compound cases. For example, you can change a CAD assembly by both revising existing component properties and adding new components.

Exporting a CAD Assembly and Generating a SimMechanics Model for the First Time

During CAD export, SimMechanics Link assigns a unique XML identifier to each CAD component. For a table summarizing the different CAD components, see “How Associativity Is Implemented” on page 2-22.

When you import the XML file and generate a SimMechanics model from it, the corresponding model components listed in the table’s second column receive these parallel identities.

Updating a Generated SimMechanics Model by Modifying CAD Assembly Properties

If you modify a CAD assembly and export a new Physical Modeling XML file from it, updating the model with the modifications allows you to reuse an existing SimMechanics model that was previously translated from the same assembly.

You modify a CAD assembly when you change the properties of its components without changing their identity. For a table of CAD assembly components you can modify, see “How Associativity Is Implemented” on page 2-22.

You update a generated SimMechanics model when you import the Physical Modeling XML file for the modified CAD assembly. The updated model reflects the new component properties in the modified assembly.

Associativity identifies the components in the existing generated SimMechanics model so that the importer can update their properties.

Updating a Generated SimMechanics Model by Extending the CAD Assembly

If you add more components to a CAD assembly and export a new Physical Modeling XML file from it, updating the model with the extensions allows you to reuse an existing SimMechanics model previously translated from the same CAD assembly.

You extend a CAD assembly when you add one or more components to it. For a table of CAD assembly components you can add, see “How Associativity Is Implemented” on page 2-22.

You update a generated SimMechanics model when you import the Physical Modeling XML file for the extended CAD assembly. The updated model contains new blocks representing the new components in the extended assembly. Blocks representing original CAD components remain unchanged.

Associativity identifies the original components in the existing generated SimMechanics model so that the importer does not change them while adding the new associated components.

Modifying a Generated SimMechanics Model Associated with a CAD Assembly, Then Updating Its Associated Components

You can also manually add nonassociated components to an existing SimMechanics model previously generated from a CAD assembly, separately revise the assembly, then retranslate the assembly by update-importing the SimMechanics model with the revisions.

- The associated SimMechanics model components are updated with the CAD assembly revisions.
- The nonassociated SimMechanics model components are not unchanged.
- If the nonassociated SimMechanics model components are connected in the original model to associated blocks, they might become disconnected after update-import.

- Nonassociated model components can include Constraints, Drivers, Actuators, and Sensors that you manually added and connected to associated, imported Bodies and Joints.
- Nonassociated model components can also include Bodies and Joints added manually after you generated the original SimMechanics model. These Bodies and Joints were not import-generated and therefore cannot be associated.

Troubleshoot CAD Export Issues

In this section...
“Unsupported Constraints” on page 2-27
“Use SimMechanics Link Help” on page 2-28

During CAD export, SimMechanics Link can encounter CAD translation issues that arise from unsupported CAD elements. The following issues are known.

Unsupported Constraints

If a CAD assembly contains one or more unsupported constraints or mates, the SimMechanics Link utility issues a warning message and creates a log file in the folder of the XML import file. The warning message identifies the name of the log file; the log file identifies the unsupported CAD constraints. In the event a constraint warning occurs, review the log file and consider replacing the unsupported constraints or mates with supported equivalents.

The constraint warning does not stop the creation of the XML import file. You can import the XML file into a new SimMechanics model, but the model does not accurately represent the original CAD assembly. During import, constraint warnings appear at the MATLAB command line.

For a list of supported constraints, see “Creo Constraint – SimMechanics Joint Correspondence”.

Correct Constraint Issue

When you import a CAD assembly that contains an unsupported constraint, SimMechanics replaces the constraint with a Weld Joint block. The block provides zero degrees of freedom between the connecting rigid body frames. To restore the original degrees of freedom between the rigid bodies frames, you must:

- In the CAD platform, replace each unsupported constraint with a supported equivalent. When you are finished, export the CAD assembly once again.

- In the imported SimMechanics model, manually replace each Weld Joint replacement block with the appropriate joint block.

Note Replacing unsupported constraints with supported equivalents works in a limited number of cases. In certain cases, no adequate equivalent constraint or joint block exists.

Use SimMechanics Link Help

In the CAD platform, the SimMechanics Link utility provides a **Help** option. Select the **Help** option to open the online SimMechanics Link help in MATLAB.

First-Generation Examples

Using realistic examples, these case studies illustrate how to translate mechanical systems that are defined externally, as computer-aided design (CAD) assemblies, into SimMechanics models.

- “Case Study Overview” on page 3-2
- “Export a CAD Rigid Body” on page 3-4
- “Translate CAD Constraints into SimMechanics Joints” on page 3-7
- “Export and Re-Export a Pendulum Assembly” on page 3-20
- “Export CAD Robot Arm” on page 3-40
- “Export a CAD Stewart Platform” on page 3-43

Case Study Overview

The case studies show how to export a computer-aided design (CAD) assembly into a SimMechanics model using the SimMechanics Link utility. Case studies include:

- “Export a CAD Rigid Body” on page 3-4 — Translate CAD assembly with a single part into a SimMechanics model with a single rigid body.
- “Translate CAD Constraints into SimMechanics Joints” on page 3-7 — Translate CAD assemblies with two constrained parts into SimMechanics models with two rigid bodies connected by a joint.
- “Export and Re-Export a Pendulum Assembly” on page 3-20 — Modify a CAD assembly and update the imported SimMechanics model.

Note This study requires Pro/ENGINEER.

- “Export CAD Robot Arm” on page 3-40 — Translate an intermediate CAD assembly containing a simple subassembly hierarchy.
- “Export a CAD Stewart Platform” on page 3-43 — Translate a advanced CAD assembly with a complex subassembly hierarchy.

Note This study is not supported for Autodesk® Inventor.

Requirements

The following assembly and exporting examples require a CAD platform supported by the SimMechanics Link utility. To complete all the steps, you also need SimMechanics Link and SimMechanics software.

The example CAD files are under *\$matlabroot/toolbox/physmod/smlink/smlinkdemos/*, in the subfolder appropriate to your supported CAD platform. The file extensions of these files are specific to each CAD platform.

The assembly, geometric, kinematic, and part details differ from platform to platform. In SolidWorks, constraints on CAD parts are called *mates*.

For More About Simulink and SimMechanics Blocks and Functions

For more information about SimMechanics software, consult the block and command reference of the SimMechanics documentation.

For more information about Simulink software, consult the block and function reference of the Simulink documentation.

Export a CAD Rigid Body

In this section...
“Access the Example Files” on page 3-4
“Examine Rigid Body Parameters” on page 3-4
“Export CAD Rigid Body” on page 3-6
“Import CAD Rigid Body” on page 3-6

Access the Example Files

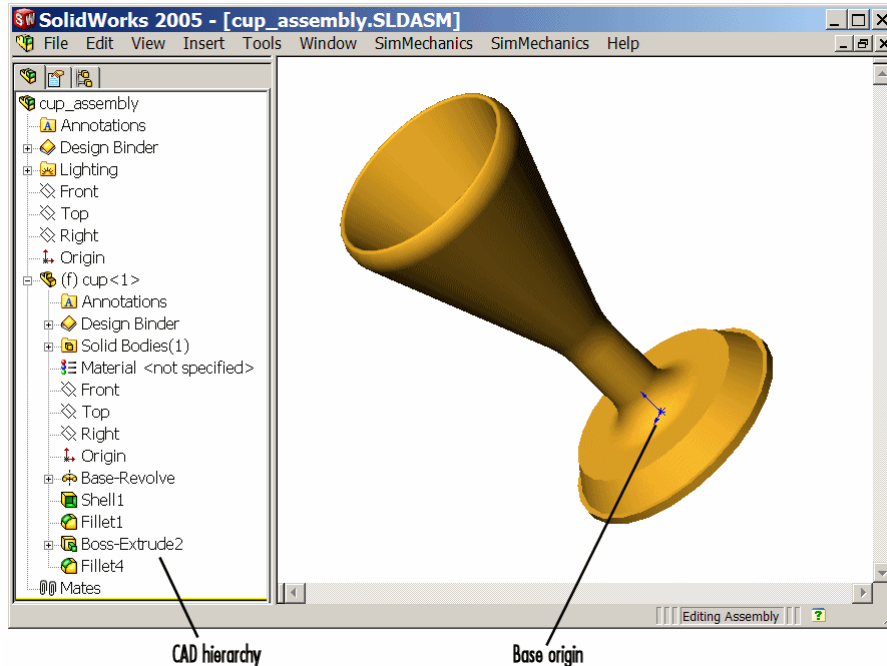
In this example, you export an assembly with one part and no constraints. Look for the following two example CAD files in the SimMechanics Link examples folder:

- The full assembly file, `cup_assembly.ASSEMBLYFILETYPE`
- The part, a cup, in a file called `cup.PARTFILETYPE`

Although it has only one part, you must export the full assembly into XML, not just the cup part.

Examine Rigid Body Parameters

Open the cup assembly file in your CAD platform and check its geometry and mass properties.



Cup Assembly in a CAD Platform

Property	Value
Volume	0.0001 cubic meters (m ³)
Surface area	0.0381 square meters (m ²)
Density	3.0 grams/cm ³ = 3000 kg/m ³
Mass	0.2906 kilograms (kg)
Principal moments of inertia at the center of gravity	$I_x = 0.00015$, $I_y = 0.00067$, $I_z = 0.00067$ kg-m ²

The inertia tensor is computed at the center of gravity, with the coordinate axes aligned with assembly base-origin axes, indicated in the figure. The x -axis is the cup's axis of symmetry, and the y - and z -axes point across the cup.

Export CAD Rigid Body

Using the SimMechanics Link interface to your CAD platform, export the assembly into Physical Modeling XML format. The XML file `cup_assembly.xml` appears in your working CAD folder.

Import CAD Rigid Body

Once you have generated the Physical Modeling XML file, you are ready to import the cup assembly into a SimMechanics model. For a step-by-step description of the CAD Import procedure, see “Import a CAD Rigid Body”.

Translate CAD Constraints into SimMechanics Joints

In this section...

“Access the Example Files” on page 3-8

“Translation of Two-Part Models” on page 3-8

“Block Structure of Two-Part Models” on page 3-9

“Translate Assembly with Six-DoF Joint” on page 3-10

“Translate Assembly with Prismatic Joint” on page 3-11

“Translate Assembly with Revolute Joint” on page 3-15

“Translate Assembly with Rectangular Joint” on page 3-16

“Translate Assembly with Spherical-Spherical Massless Connector” on page 3-17

In “Export a CAD Rigid Body” on page 3-4, you create and export an assembly composed of a single part. Because there are no other parts in that CAD assembly, the SimMechanics body is welded to ground and has no degrees of freedom (DoFs). This lack of DoFs is not realistic for most assemblies.

This study presents a set of complete CAD assemblies with both parts and constraints and possessing DoFs. Each example assembly consists of two instances of the same part file, representing two identical cubes. The study shows how to find the required files, presents the essential steps for generating models from them, and describes the structure common to all the generated models. It ends with specific assembly cases that include two cubes:

- With no constraints, so that the cubes have the full six degrees of freedom relative to one another
- Constrained in two different ways so as to produce the same result, a single prismatic (translational) DoF between them
- Constrained so as to allow only a single revolute (rotational) DoF between them
- Constrained so as to allow two prismatic (translational) DoFs between them
- Constrained so as to allow relative spherical joint motion, with the two cubes separated by a constant nonzero distance

In different assemblies, the two cubes are constrained with different constraint combinations to create different relative DoFs between the cubes. In most cases, you can represent one set of DoFs with a large number of different combinations of constraints.

Access the Example Files

Look for the CAD assembly files of this study in the *smlink/smlinkdemos* directory. The assemblies have the generic name, <assembly-name>.ASSEMBLYFILETYPE. The cube part is in *magic_cube.PARTFILETYPE*.

Assembly Name	Assembly Configuration
sixDOF	Two cubes with no constraints
prismatic1	Two cubes with planar and cylindrical constraints
prismatic2	Two cubes with planar constraints
revolute	Two cubes with planar and cylindrical constraints
inplane	Two cubes with planar constraints
spherical_spherical_massless_connector	Two cubes with a distance constraint

Translation of Two-Part Models

The procedure for exporting a two-part assembly and generating SimMechanics models based on it is essentially the same for all the examples of this study.

Export Assembly

To see a two-part assembly and export it into a Physical Modeling XML file:

- 1 Open the assembly, <assembly-name>.ASSEMBLYFILETYPE. The two parts are *magic_cube-1* and *magic_cube-2*.

Locate any constraints imposed on the parts. These constraints define the relative DoFs between the parts.

- 2 Using the SimMechanics Link interface to your CAD platform, export the CAD assembly into Physical Modeling XML. The XML file is saved in your current working CAD folder.

Import Model

You can import the Physical Modeling XML file into SimMechanics. SimMechanics interprets the file and generates a corresponding SimMechanics model. The new model is based on XML file structure and parameters. The CAD Import procedure requires access to SimMechanics.

- 1 Move or copy the XML file to a working MATLAB folder. In this folder, open MATLAB.
- 2 At the command line, enter `mech_import('<assembly_name>')` to automatically generate a model, `<assembly_name>.mdl`, based on `<assembly_name>.xml`.
- 3 Open the subsystem. The blocks are arranged in the common structure described in “Block Structure of Two-Part Models” on page 3-9. A set of Joints represents the DoFs between the two cubes.

Block Structure of Two-Part Models

In this study, all the models that you generate each have eight blocks. From the example CAD assemblies, the models have a common structure because each assembly has a fundamental root and two moving parts:

- *The assembly fundamental root.* As in any generated CAD-based model, the four-block combination Machine Environment – Root Ground – Root Weld – Root Part represents the assembly fundamental root. The Root Part is a nonmoving, zero-mass/zero-inertia body.
- *The moving bodies.* The bodies representing the assembly parts are `magic_cube-1` and `magic_cube-2`.
- *The joints.* In all the models, the first cube is connected by a Weld to Root Part and cannot move. The second cube is connected to RootPart by a Joint that represents the appropriate degrees of freedom (DoFs).

Depending on the DoFs in a particular assembly, CAD translation configures the Joint to represent different DoFs with combinations of prismatic, revolute, and spherical primitives. The second cube can move with respect to the first cube through the DoFs represented by the Joint.

Some of the blocks in the generated models are redundant. You can manually edit and simplify the models without changing their physical properties.

CAD Constraints Restrict Degrees of Freedom

CAD platforms treat unconstrained parts as free rigid bodies with six mechanical degrees of freedom. To reduce the degrees of freedom of a CAD part, and therefore the ways that part can move, you must *constrain* that part. Applying a CAD constraint between two parts reduces the ways the two parts can move with respect to each other. In a two-part CAD assembly, one part is static relative to ground.

Translate Assembly with Six-DoF Joint

The simplest assembly with two parts has no constraints between the parts. The parts can move with respect to one another with all six degrees of freedom (DoFs).

Export Assembly

To see and export the assembly:

- 1 Open the assembly, `sixDOF.ASSEMBLYFILETYPE`.

The cube parts have no constraints. Therefore, relative to one another, the cubes are unconstrained in their motion and have six relative DoFs.

- 2 Export this CAD assembly into the file, `sixDOF.xml`.

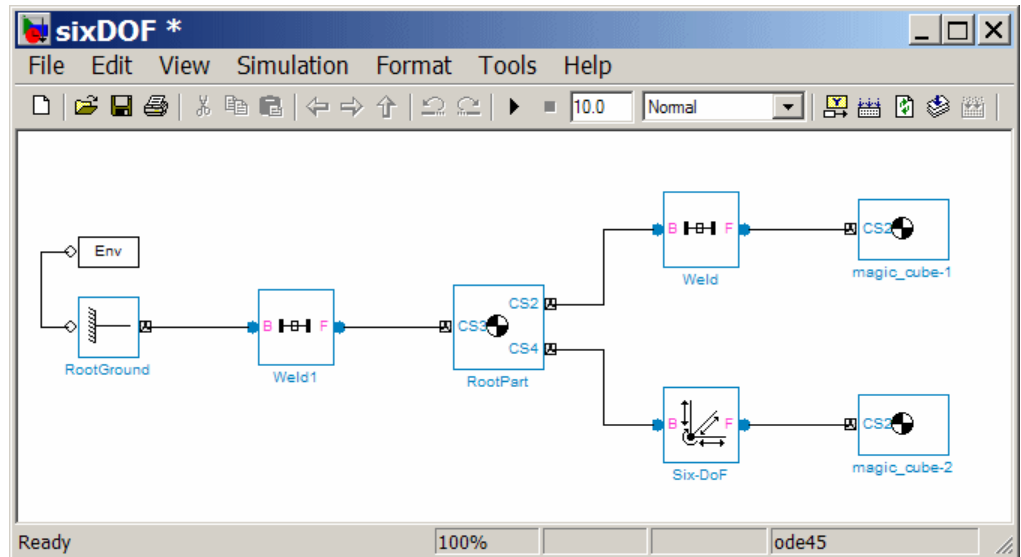
Import Model

To generate a model based on the assembly:

- 1 At the MATLAB command line, enter `mech_import('sixDOF')` to generate a model, `sixDOF.mdl`.

- 2 Inspect the model. There are eight blocks.

The Six-DoF Joint represents the six DoFs between the two cubes with one spherical and three prismatic primitives.



Translate Assembly with Prismatic Joint

In the following two assemblies, the two cubes are constrained to have only a single translational degree of freedom (DoF) between them. These assemblies illustrate two ways to express one translational degree of freedom. You can experiment with other constraints to find more. In the translated SimMechanics models, this single DoF is a prismatic joint.

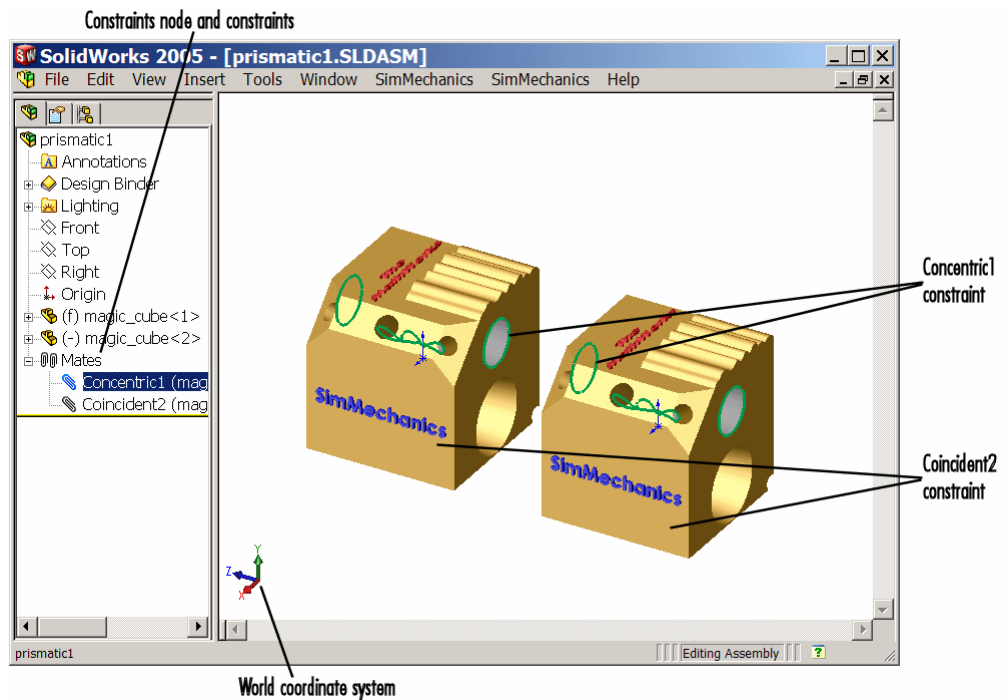
Prismatic as a Cylindrical Constraint and a Planar Constraint

To see the first way of constraining the DoFs to produce a prismatic joint:

- 1 Open the assembly file, `prismatic1.ASEMBLYFILETYPE`, and examine its CAD hierarchy.
- 2 Locate the two constraints on the two cubes.

- Highlight the cylindrical constraint in the assembly. This constraint allows the two cubes to only slide along and rotate about the z -axis running through the center of the parallel and concentric upper holes of each cube.
- Highlight the planar constraint in the assembly. This constraint allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane, representing two translational DoFs. The constraint also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.

With these two constraints, the two cubes can only slide along the z -axis common to the two upper concentric holes. The second constraint prevents rotation about this axis, leaving the whole assembly with only one translational DoF.



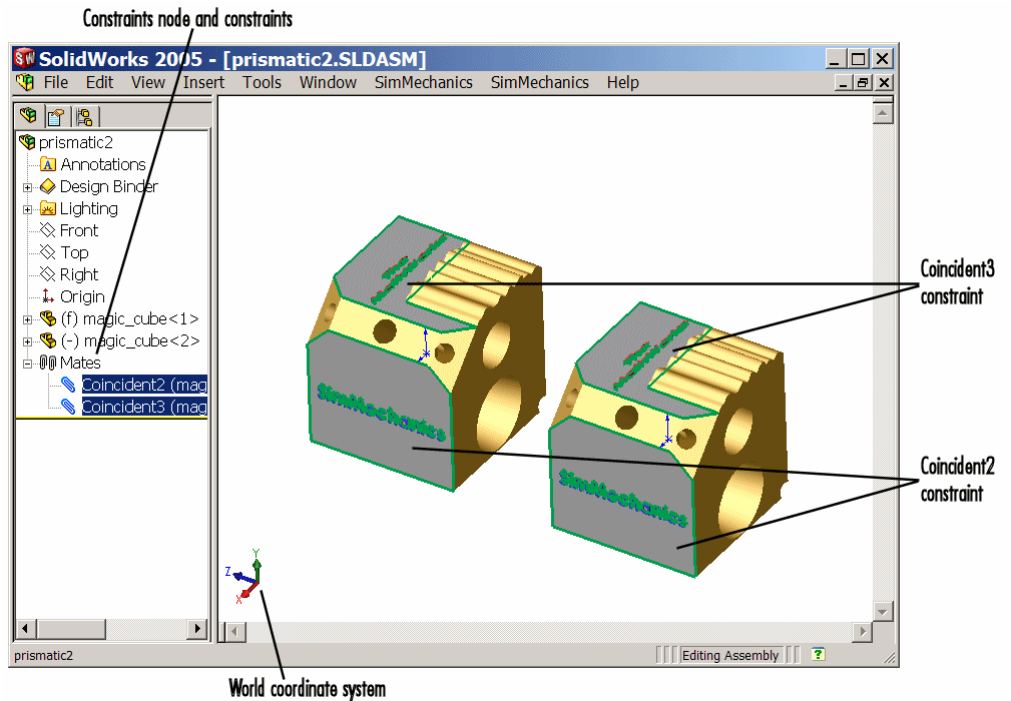
Cylindrical and Planar Constraints on Two Cubes (SolidWorks®)

Prismatic as Two Orthogonal Planar Constraints

To see the second way of constraining the DoFs to produce a prismatic joint:

- 1 Open the assembly file, `prismatic2.ASSEMBLYFILETYPE`, and examine its CAD hierarchy.
- 2 Locate the two constraints on the two cubes.
 - Highlight the first planar constraint in the assembly. This constraint allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane, representing two translational DoFs. It also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.
 - Highlight the second planar constraint in the assembly. This constraint allows the two cubes to slide along the x - z plane, with the two sides marked “The MathWorks” sharing a common plane, representing two translational DoFs. The constraint also allows the two cubes to rotate about the y -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the x - z plane.

With these two constraints, the two cubes can only slide along the z -axis common to the two planes y - z and x - z , leaving the whole assembly with only one translational DoF.



Two Planar Constraints on Two Cubes (SolidWorks®)

Export Assemblies and Import Models

To create models from the assemblies:

- 1 Export the two assemblies into the XML files, `prismatic1.xml` and `prismatic2.xml`.
- 2 Copy or move them to a MATLAB working folder. At the MATLAB command line, generate SimMechanics models using `meh_import`.

In both models, the assemblies are translated into block diagrams of eight blocks each. The Prismatic Joint represents the single translational DoF between the two cubes with one prismatic primitive along the z -axis.

Translate Assembly with Revolute Joint

In the following assembly, the two cubes are constrained to have only a single rotational degree of freedom (DoF) between them. In the translated SimMechanics model, this single DoF is a revolute joint.

Open CAD Assembly

To see an assembly with one rotational DoF:

- 1 Open the assembly file, `revolute.ASSEMBLYFILETYPE`, and examine its CAD hierarchy.
- 2 Locate the two constraints on the two cubes.
 - Highlight the cylindrical constraint in the assembly. This constraint allows the two cubes to slide along and rotate about the z -axis running through the center of the parallel and concentric upper holes of each cube.
 - Highlight the planar constraint in the assembly. This constraint allows the two cubes to slide along the x - y plane, with the parallel sides sharing a common plane. The constraint also allows the two cubes to rotate about the z -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the x - y plane.

With these two constraints, the two cubes can only rotate about the z -axis orthogonal to the x - y plane, leaving the whole assembly with only one rotational DoF.

Export Assembly and Import Model

To generate a model based on the assembly:

- 1 Export the assembly as the file, `revolute.xml`. Copy or move it to a MATLAB working folder.
- 2 At the MATLAB command line, generate a SimMechanics model using `mech_import`.

The assembly is translated into a block diagram of eight blocks. The Revolute Joint represents the single rotational DoF between the two cubes with one revolute primitive about the z -axis.

Translate Assembly with Rectangular Joint

In the following assembly, the two cubes are constrained to have only two translational degrees of freedom (DoFs) between them. In the translated SimMechanics model, these two DoFs are two prismatic joints.

Open CAD Assembly

To see an assembly with two translational DoFs:

- 1 Open the assembly file, `inplane.ASSEMBLYFILETYPE`, and examine its CAD hierarchy.
- 2 Locate the two constraints on the two cubes.
 - Highlight the first planar constraint in the assembly. This constraint allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane. The constraint also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.
 - Highlight the second planar constraint in the assembly. This constraint allows the two cubes to slide parallel to the x - z plane, with the two sides marked “The MathWorks” parallel but not necessarily in the same plane. It also allows the two cubes to translate perpendicular to the x - z plane and to rotate about the y -axis. The cubes are not allowed to rotate about any other axis.

With these two constraint, the two cubes can only slide in the y - z plane, leaving the whole assembly with only two translational DoFs.

Export Assembly and Import Model

To generate a model based on the assembly:

- 1 Export the assembly as the file, `inplane.xml`. Copy or move it to a MATLAB working folder.

- 2 At the MATLAB command line, generate a SimMechanics model using `mech_import`.

The assembly is translated into a block diagram of eight blocks. The In-Plane Joint represents the two translational DoFs between the two cubes with two prismatic primitives, along the y -axis and the z -axis.

Translate Assembly with Spherical-Spherical Massless Connector

In the following assembly, the two cubes are constrained to have six rotational degrees of freedom (DoFs) between them, represented by two spherical primitives. The spherical primitives pivot independently about two pivot points at a fixed relative distance. In the translated SimMechanics model, a spherical-spherical massless connector represents these six DoFs.

Open CAD Assembly

To see an assembly with three rotational DoFs separated from three other rotational DoFs:

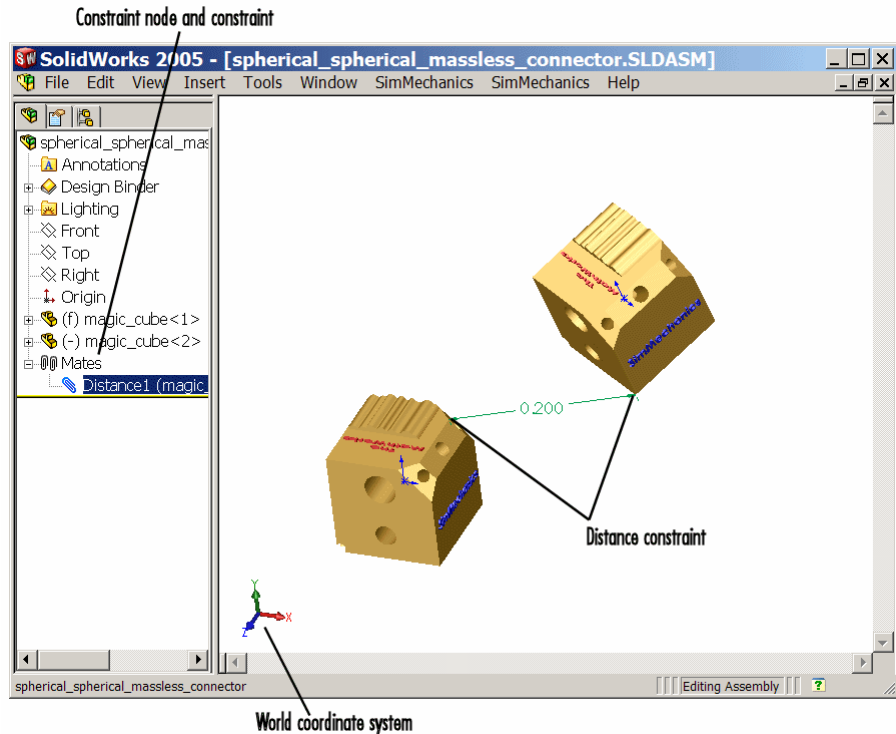
- 1 Open the assembly file,

```
spherical_spherical_massless_connector.ASSEMBLYFILETYPE
```

and examine its CAD hierarchy.

- 2 Locate the one constraint on the two cubes.

Highlight this distance-offset constraint. The two spherical pivot points occur one on each cube and mark the endpoints of the rigid massless connector. The cubes can move so that the distance between these two endpoints (the length of the massless connector) does not change. The constraint allows the two cubes to pivot independently about their connector endpoints.



Distance Constraint on Two Cubes (SolidWorks®)

Export Assembly and Import Model

To generate a model based on the assembly:

- 1 Export the assembly as a file,

```
spherical_spherical_massless_connector.xml
```

- 2 Copy or move it to a MATLAB working folder.

- 3 At the MATLAB command line, generate a SimMechanics model using `mech_import`.

The assembly is translated into a block diagram of eight blocks, arranged in the common structure described in “Block Structure of Two-Part Models” on page 3-9.

The Spherical-Spherical massless connector Joint block represents the two spherical primitives, each with three rotational DoFs, independently pivoting at each end of the massless, rigid connector connecting the two cubes.

Export and Re-Export a Pendulum Assembly

In this section...
“About Model Update” on page 3-20
“CAD Assembly Files” on page 3-20
“Translate CAD Assembly” on page 3-21
“Modify Pendulum Assembly and Update SimMechanics Model” on page 3-26
“Add New Body to Create a Triple Pendulum” on page 3-32
“Update Imported Model While Retaining Manual Joint Replacements” on page 3-37
“Selectively Update Imported Model” on page 3-38

About Model Update

Note The CAD assembly files of this case study require Pro/ENGINEER.

The following example shows how to update a previously generated SimMechanics model with a new XML file exported after changes have been made to the original CAD assembly. The example starts with a double pendulum assembly.

For a general discussion of retranslation and update-import, see “Retranslate CAD Assembly” on page 2-21.

CAD Assembly Files

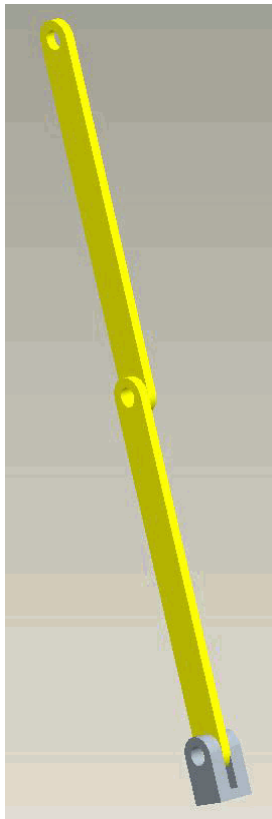
Look for the following seven CAD files of this case study in the SimMechanics Link examples folder.

File Name	CAD File Type
dpen.ASSEMBLYFILETYPE	Assembly
tpen.ASSEMBLYFILETYPE	Assembly

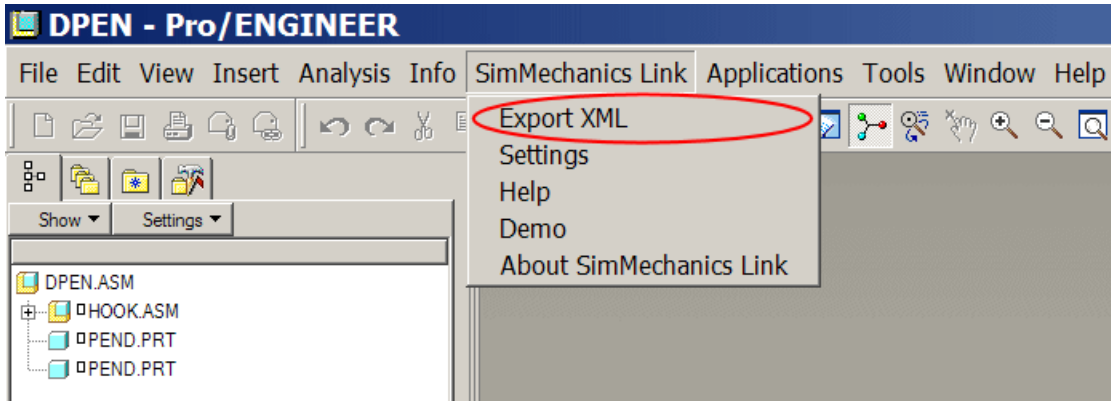
File Name	CAD File Type
hook. <i>ASSEMBLYFILETYPE</i>	Subassembly
mid_part. <i>PARTFILETYPE</i> pend. <i>PARTFILETYPE</i>	Parts (main assembly)
hook. <i>PARTFILETYPE</i> hookbase. <i>PARTFILETYPE</i>	Parts (subassembly)

Translate CAD Assembly

Open the dpen assembly file and export it into an XML file.



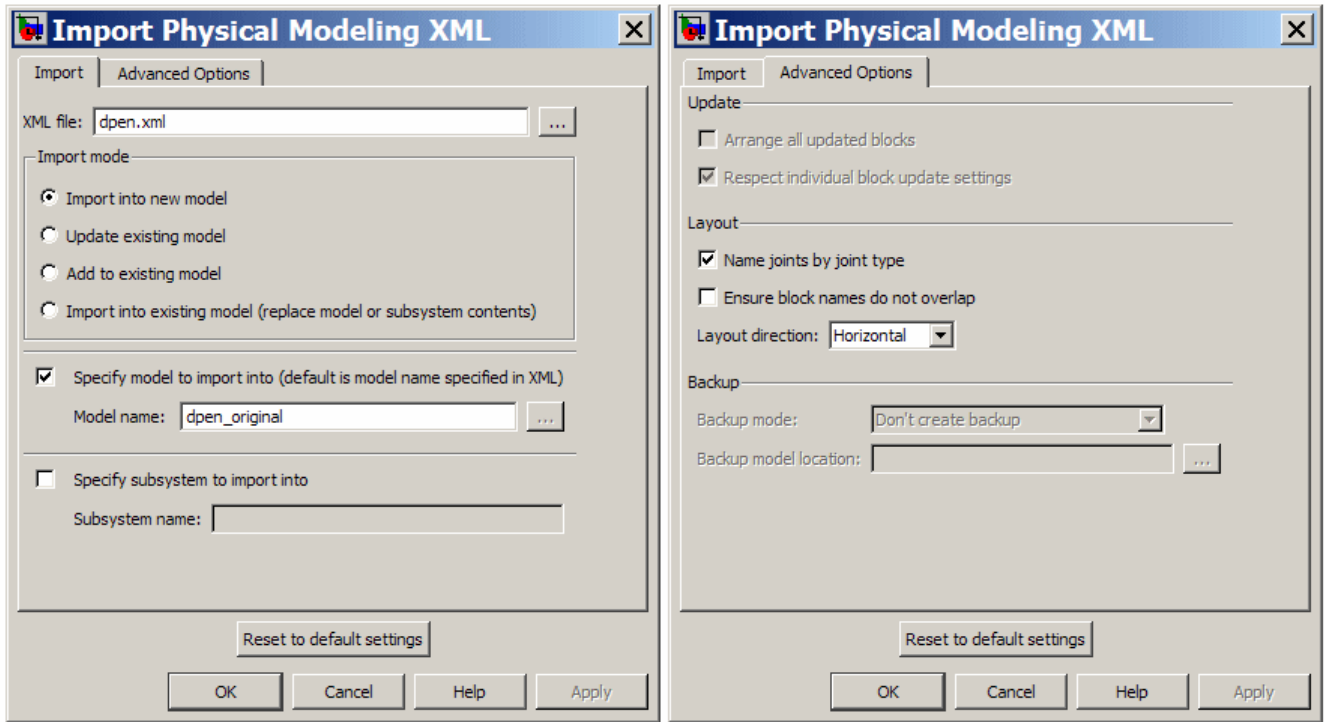
Export the assembly through the **SimMechanics Link** menu of the CAD platform's menu bar. The result is an XML file called `dpen.xml`.



Import CAD Assembly and Generate SimMechanics Model

To import the XML file and generate the SimMechanics model, you can either use the import dialog box or work with the full command at the MATLAB command line.

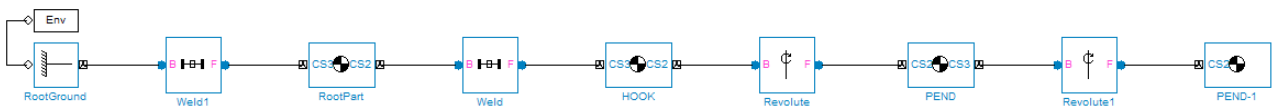
Open the dialog box by entering `mech_import` at the command line.



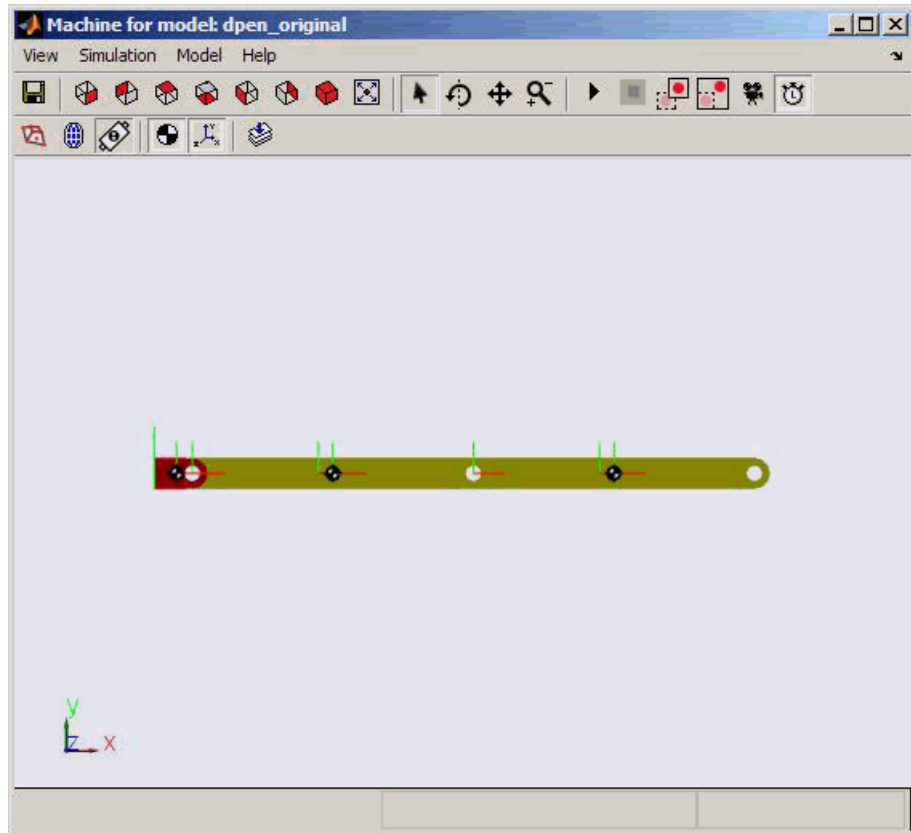
The alternative is to use the full command with specified import options:

```
mech_import('dpen.xml', 'ImportMode', 0, ...
           'ModelToImportInto', 'dpen_original');
```

This model results from the import step. The blocks are created at the top level.



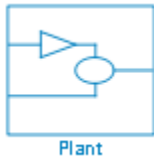
With visualization enabled, you see this SimMechanics model with custom bodies defined by STL files exported when the XML file was exported.



Import Physical Modeling XML File

If you want to import the assembly into a model subsystem, rather than at the top level of a model, use the `SubsystemToImportInto` option of the `mech_import` command. (You can also use the parallel options in the import dialog box.) For example, you can import the assembly into a subsystem named `Plant`.

```
mech_import('dpen.xml','ImportMode',0,...  
           'ModelToImportInto','dpen_original',...  
           'SubsystemToImportInto','Plant');
```



Modify Pendulum Assembly and Update SimMechanics Model

You can change part properties in the original assembly and reimport the assembly, with those changes, to update the model. You can also make direct changes by hand to the original generated model and preserve those changes even as you update it with assembly changes.

In this example, you now cut out material from the links to make them lighter. You also connect joint and body sensors in the SimMechanics model to implement a controller. Now you want to update the SimMechanics model with changes from the assembly without undoing the additions you made by hand.

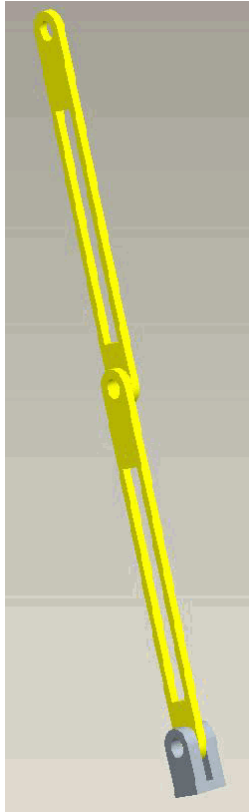
Export Modified CAD Assembly

Change the double pendulum assembly and generate a new XML file from it.

- 1 Open the pend part file and unsuppress the Extrude2 feature to enable the hole in the parts. Do this by right-clicking Extrude2 in the tree view and selecting **Resume**.

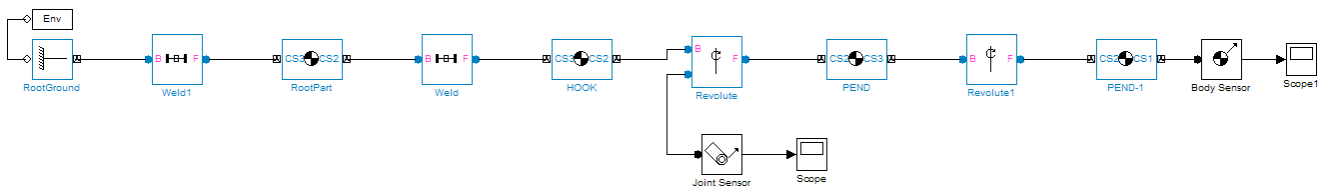
Save the part file.

- 2 Open and re-export the dpen assembly file to obtain the new XML file, which is assumed to be named `dpen_with_holes.xml`.

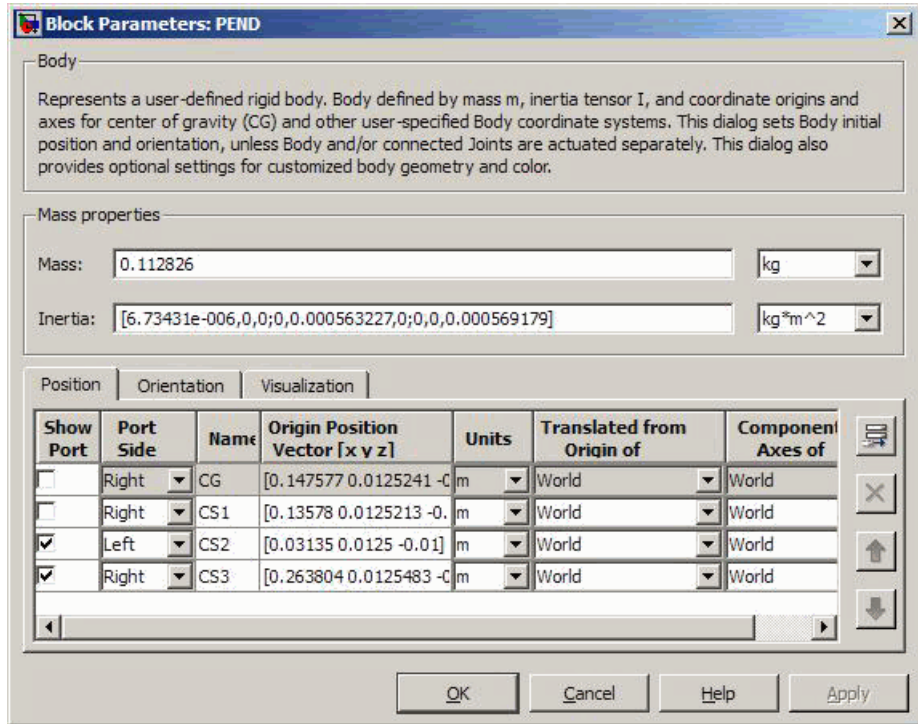


Modification to Imported SimMechanics Model

In this example, you have separately made the following changes, by hand, to the original generated model, now renamed `dpen_withsensors`. These changes include insertion of a Joint Sensor, a Body Sensor, and two Scope blocks.

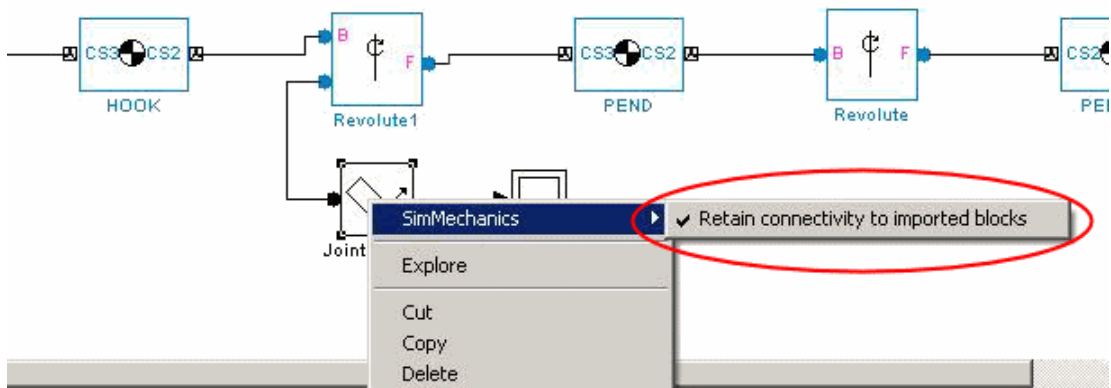


These are the properties of the PEND body before the update-import. PEND-1 has the same mass properties.



Preparation for Model Update

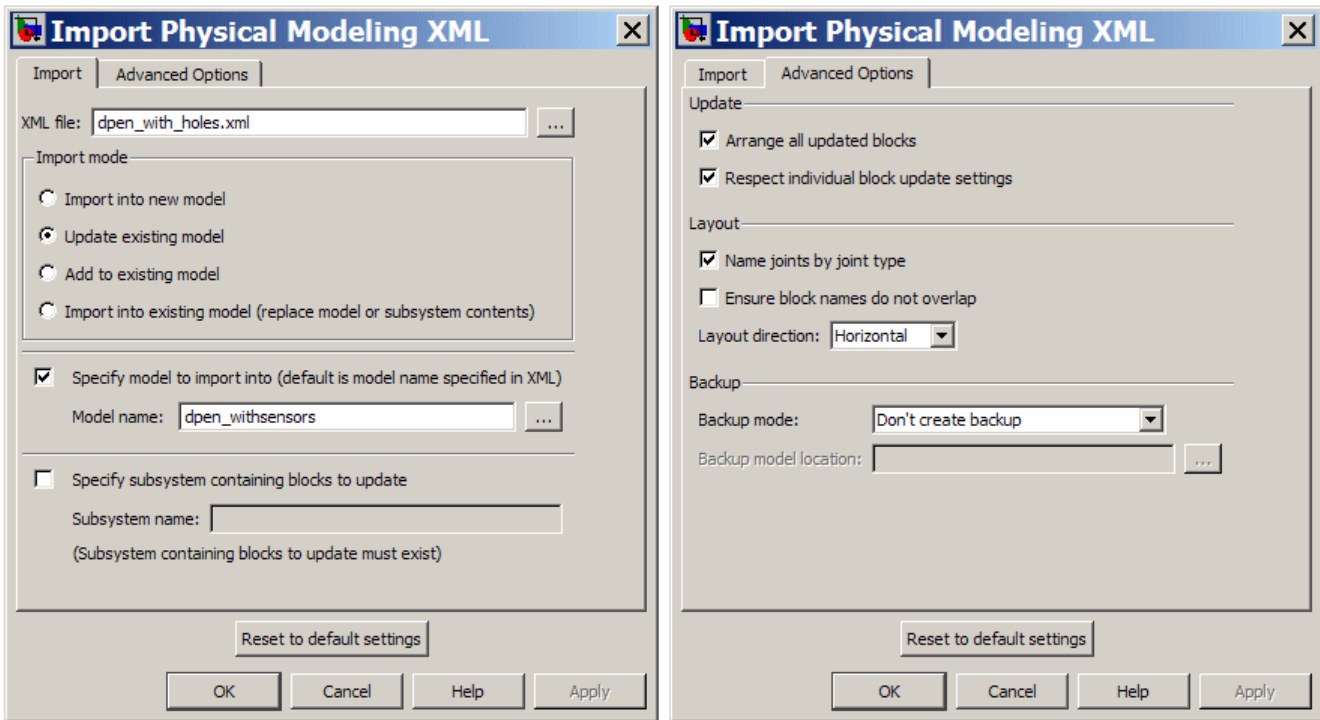
Before proceeding to the update-import from the updated CAD assembly, designate the manually added SimMechanics blocks that you want to retain and keep connected to the originally imported blocks. Right-click the sensor blocks and select the **Retain connectivity to imported blocks** option under the **SimMechanics** submenu. This option is selected by default.



SimMechanics Model Update

Now update-import in such a way as to modify the PEND and PEND-1 body mass properties, without changing the manually added blocks and connections. You can either use the import dialog box or the command line with fully specified options to update the existing model. This example uses the dialog box, opened by entering `mech_import` at the command line.

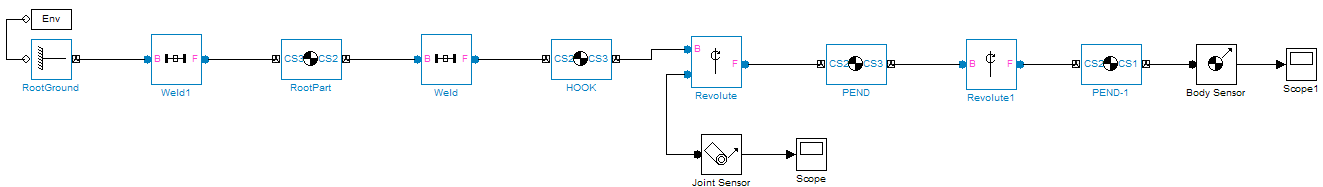
The dialog box is configured so that `dpen_with_holes.xml` is the file imported to update the modified `dpen_withsensors` model. Because the dialog box is set to update an existing model, the **Update** and **Backup** options in the **Advanced Options** tab are now enabled.



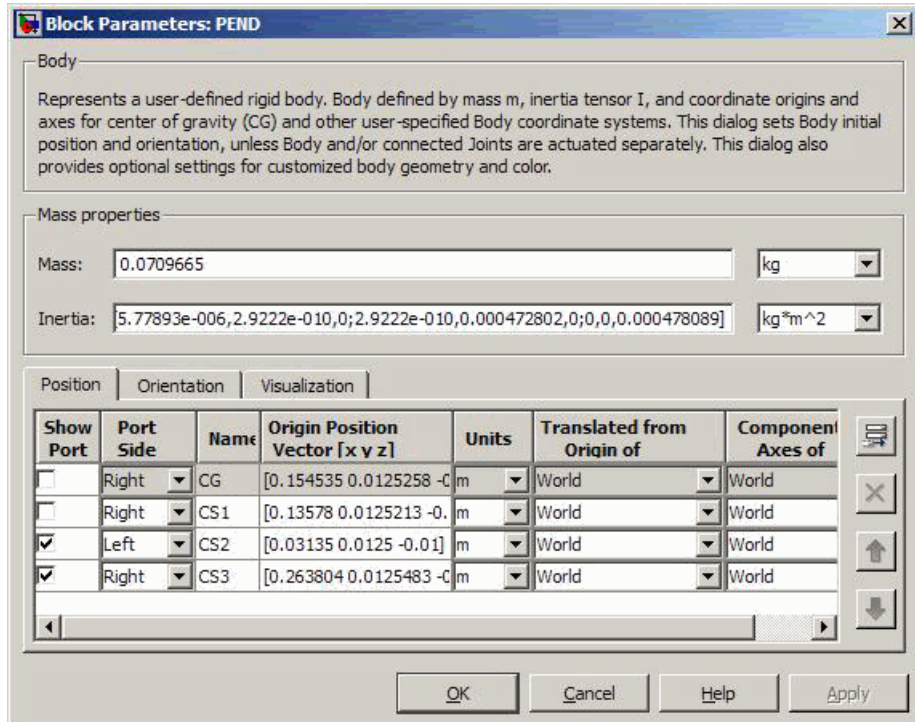
Alternatively, if you want to use the command with fully specified options, enter:

```
mech_import('dpen_with_holes','ImportMode',1,...
            'ModelToImportInto','dpen_withsensors','LayoutWithUpdate',1,...
            'EnableIndv1BlkUpdCtrl',1,'BackupMode',0)
```

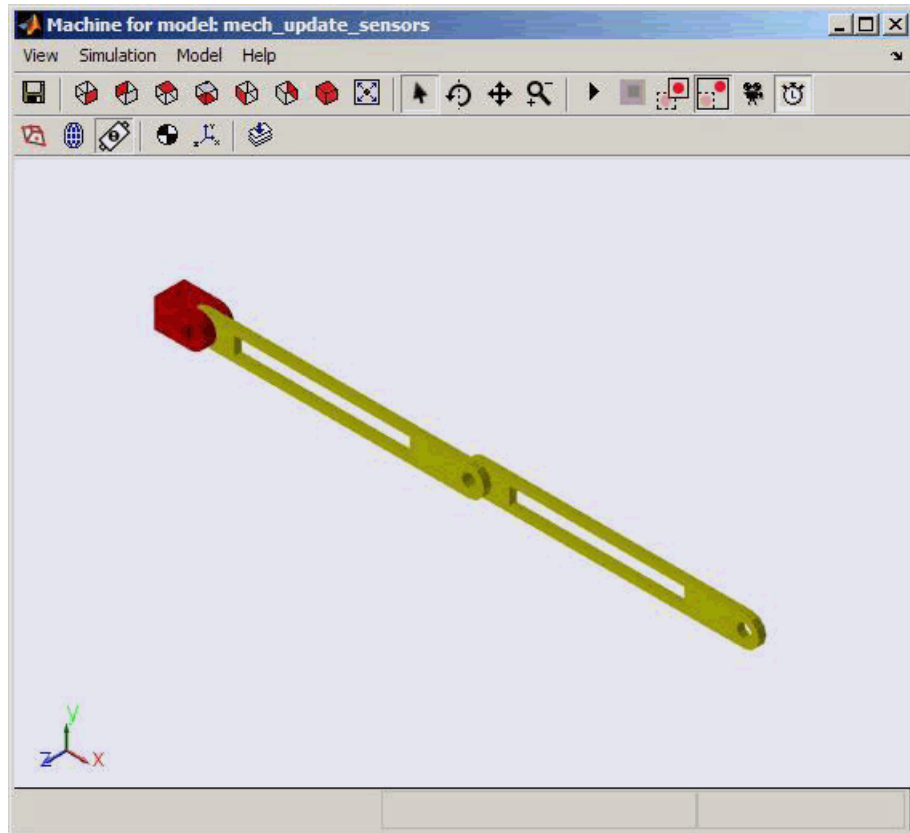
The updated model looks like the following. The sensors and Scopes are retained.



However, the PEND body properties are now different, because of the new holes in the parts. The PEND-1 mass properties are changed in the same way.



The updated bodies, displayed in SimMechanics visualization, appear differently as well, as determined by the new STL files exported from the updated assembly at the same time the new XML file was created.



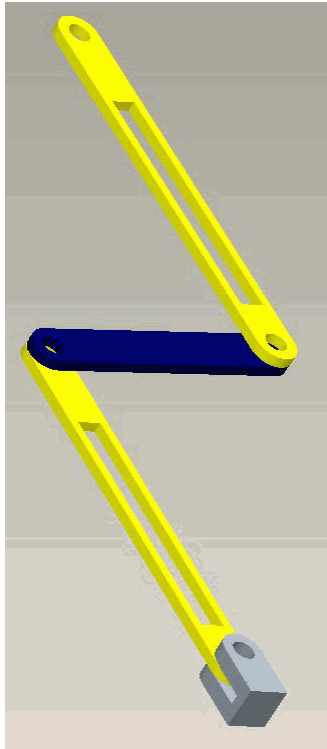
Add New Body to Create a Triple Pendulum

The changes to an assembly that you can re-export are not limited to changes in existing parts (bodies). You can add or remove parts, re-export, then update-import an older version of the generated model with the changes.

In this example, in addition to the holes in the parts you previously made, you change the original assembly further by adding a new part (link) between the two original pendulum links, making the assembly a triple pendulum. This addition changes the topology of the assembly (the assembly's connectivity), beyond the property changes to the individual links.

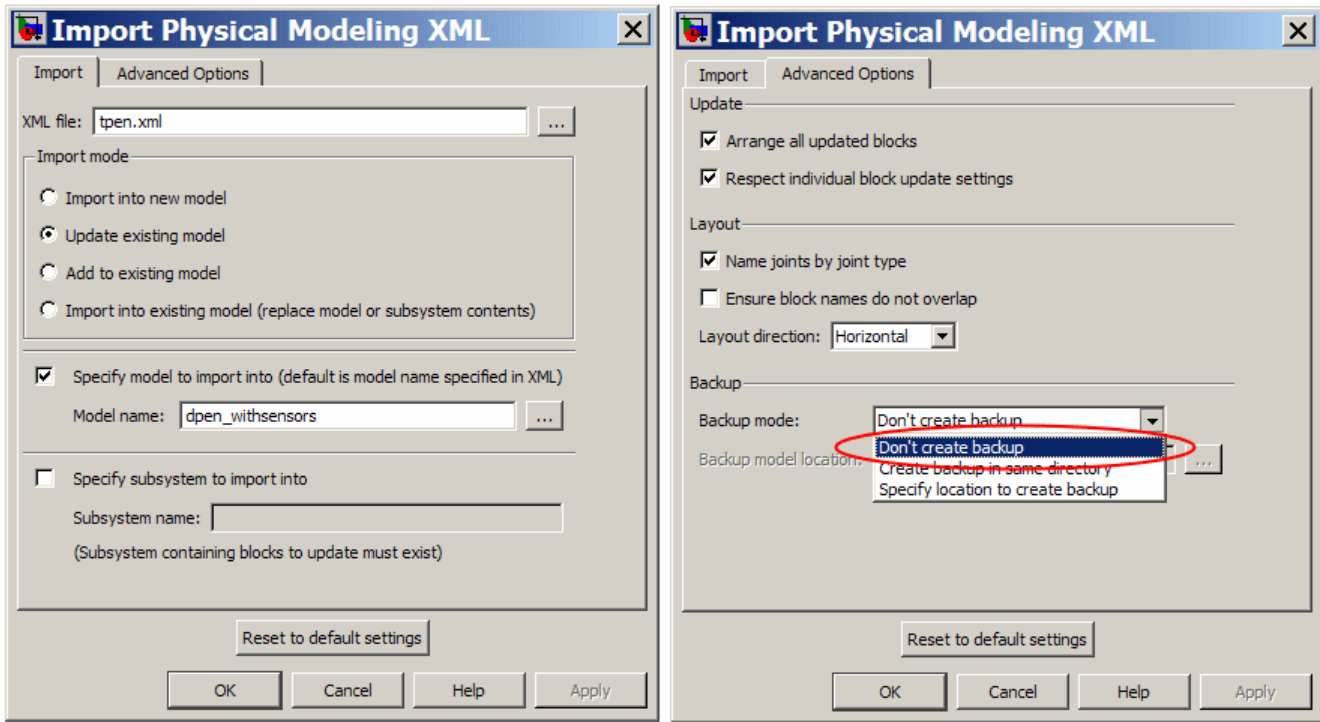
Modify and Update CAD Assembly

To see such a changed assembly, open the tpen assembly file. The tpen assembly is a copy of the dpen assembly with a link inserted. Export a new XML file, which is named tpen.xml.



Updating the Existing Generated Model

Update the existing dpen_withsensors model with the changes to the assembly. These are the required options in the import dialog box.



The **Advanced Options** tab shows important options selected for this example.

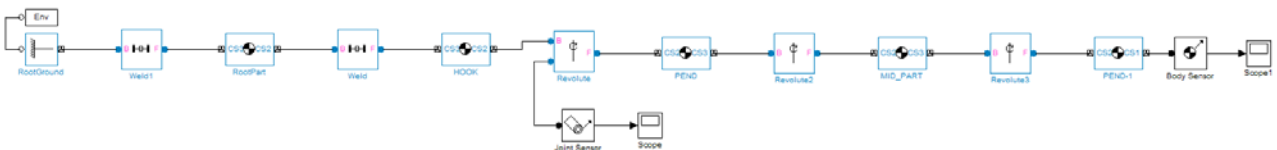
Advanced Option	Effect of Option
<p>Arrange all updated blocks</p>	<p>Selecting this option causes the importer to rearrange all the imported and updated blocks. If cleared, the importer does not alter the layout of the existing blocks, whether updated or not, and places the newly imported blocks around the bounding box for the already existing blocks. In many cases, this</p>

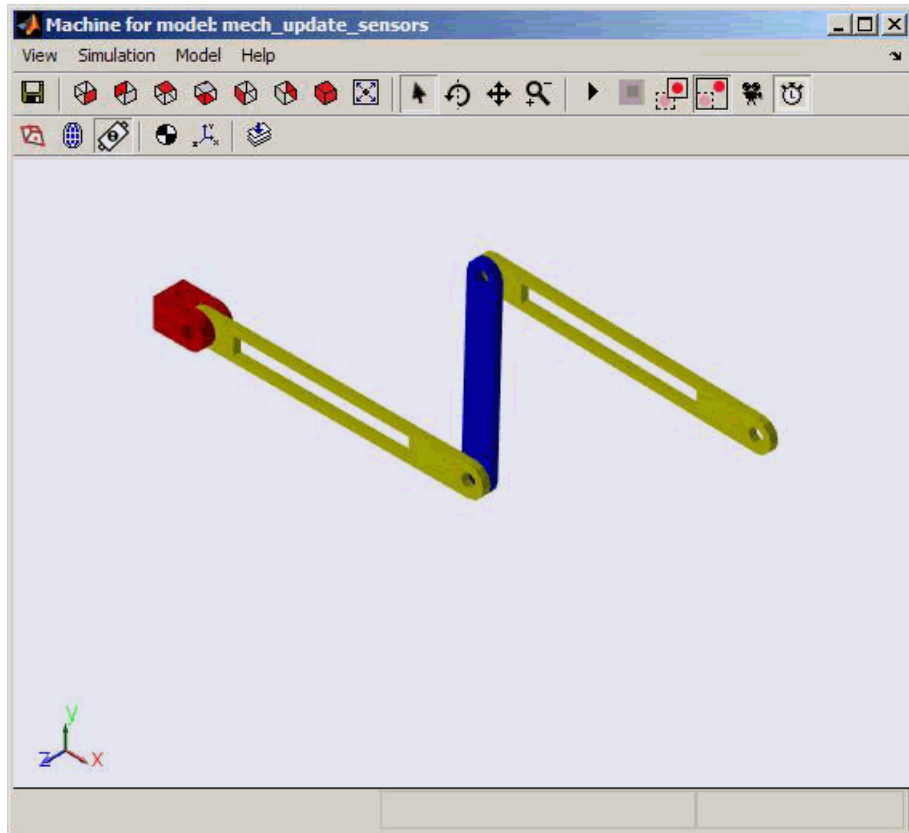
Advanced Option	Effect of Option
	means you have to rearrange the newly imported blocks by hand.
Respect individual block update settings	Selecting this option preserves the associativity information specified on each block.
Backup mode	Using this pull-down menu allows you to choose how the model is backed up during update. the backup options. By default, the backup copy of the model is created in the same folder as the model being updated.

The equivalent command line import command is:

```
mech_import('tpen.xml', 'ImportMode', 1, ...
            'ModelToImportInto', 'dpen_withsensors', 'LayoutWithUpdate', 1, ...
            'EnableIndv1BlkUpdCtrl', 1, 'BackupMode', 0)
```

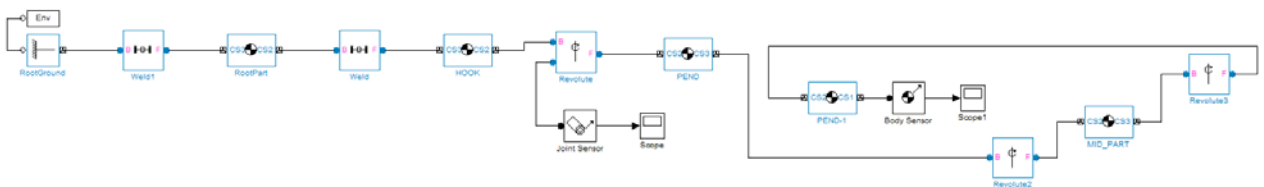
The block diagram shows that the part MID_PART inserted between the two links PEND and PEND-1. The sensors are retained. The properties of PEND and PEND-1 have been updated (to incorporate the holes that are now present). Because the **Arrange all updated blocks** option was selected, the blocks PEND and PEND-1 were moved to accommodate the block MID_PART and the joints Revolute2 and Revolute3.





Disabling Automatic Block Arrangement

The following block diagram shows the updated model that results when the **Arrange all updated blocks** option is cleared. The importer places the newly imported blocks around the bounding box of the existing blocks. The layout of the existing blocks is unchanged.



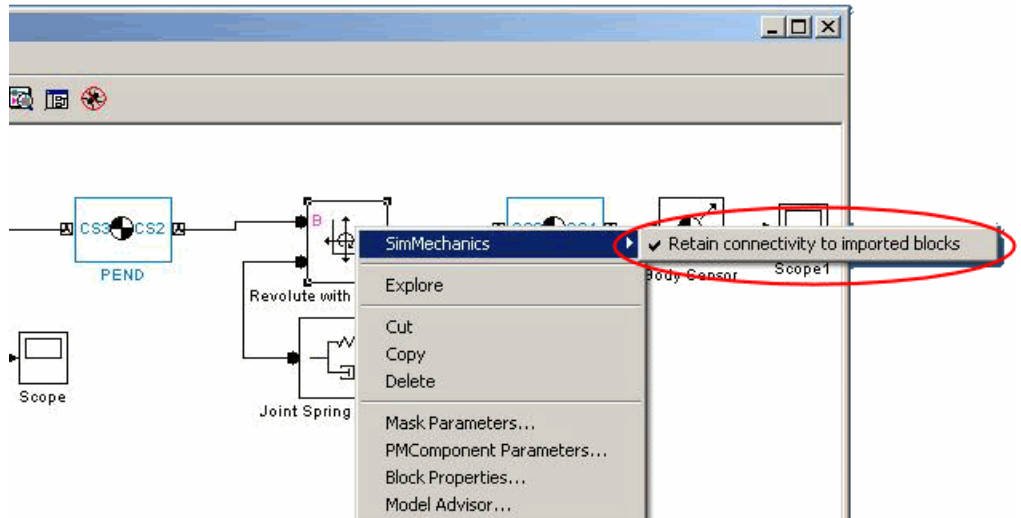
Update Imported Model While Retaining Manual Joint Replacements

In circumstances where a set of assembly constraints imposes ambiguous restrictions on the motions of two parts, the SimMechanics Link exporter cannot resolve the constraint set into a moving joint. In such cases, it exports a Weld joint. If so, it is up to you to manually replace the Weld joint with the appropriate moving joint in the generated model. Even in situations where the exporter creates a moving joint, it might not reflect the degrees of freedom that you want, and you might want replace it with another. When you have replaced such joints in an imported model, and you want to update the model with changes from the CAD assembly, you generally want to retain the replacements you made manually.

- 1 Create a `dpen_jointrep` model in the following way.

Open the `dpen_withsensors` model and replace the Revolute joint between the two link bodies by a Planar joint that has some compliance in the plane perpendicular to the original revolute axis. Rename the Planar joint to Revolute with Compliance.

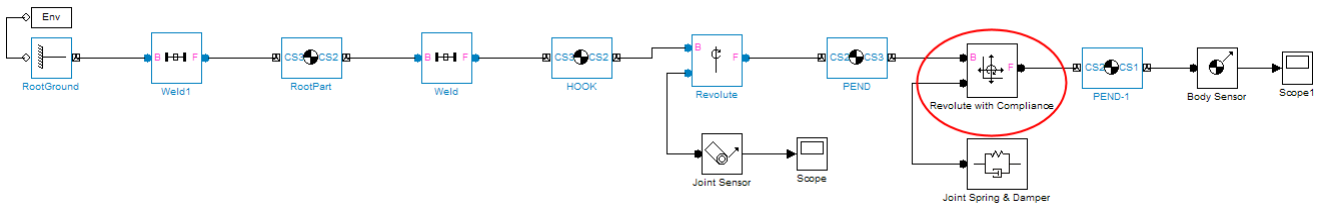
- 2 To update this model with changes to the link body properties, but to still retain the new Planar joint, right-click the Revolute with Compliance block and select the **Retain connectivity to imported blocks** option. It is selected by default.



3 Update this model with the new XML file that has changes to the properties of the links.

```
mech_import('dpen.xml', 'ImportMode', 1, ...
            'ModelToImportInto', 'dpen_jointrep', 'LayoutWithUpdate', 1, ...
            'EnableIndvBlkUpdCtrl', 1, 'BackupMode', 0)
```

The updated model retains the Revolute with Compliance joint.



Selectively Update Imported Model

You can update certain bodies while not updating other bodies with changes. Prevent a particular body from being updated during reimport by:

- 1** Right-clicking the body and selecting the **Retain without updating properties** option under the **SimMechanics** submenu.
- 2** Selecting the **Respect individual block update settings** option in the import dialog box.

If you use the full command instead, you need to set the `EnableIndv1BlkUpdCtrl` option to true.

Clearing the **Respect individual block update settings** check box causes the importer to update all blocks, disregarding the update settings on the individual blocks.

Export CAD Robot Arm

In this section...
“CAD Robot Arm Files” on page 3-40
“CAD Robot Arm Properties” on page 3-41
“Export Robot Arm Assembly” on page 3-42
“Import Model” on page 3-42

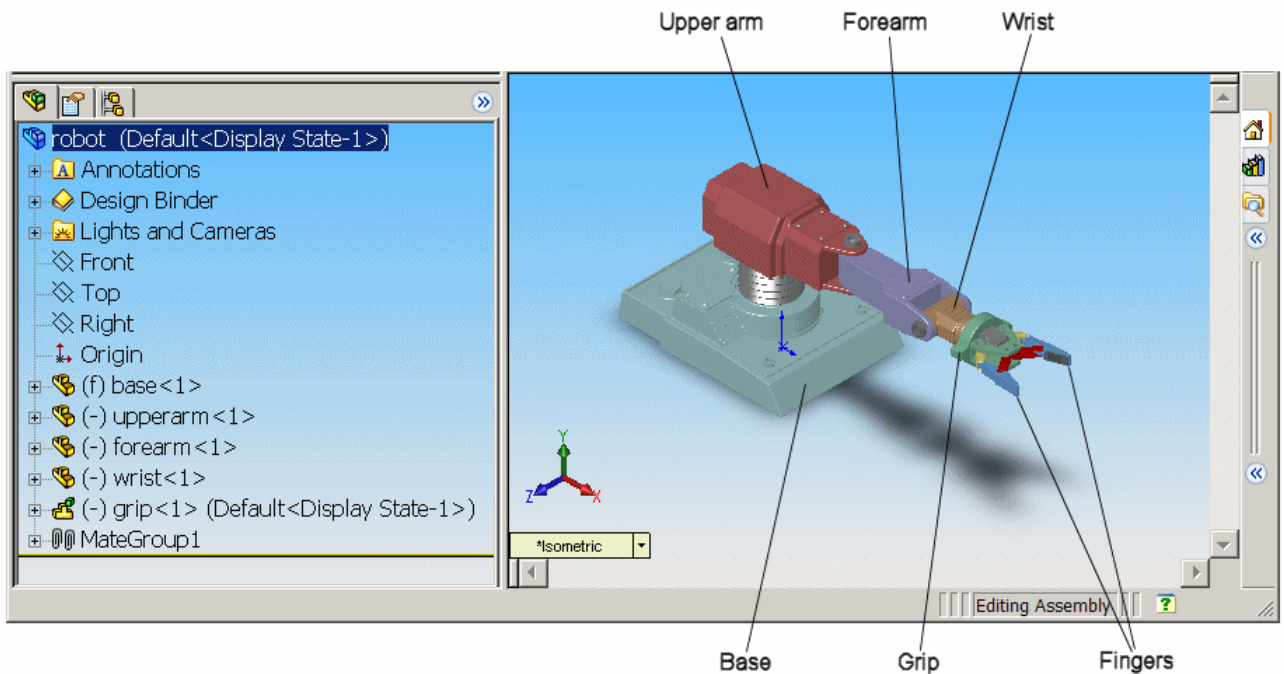
CAD Robot Arm Files

The following example is based on a more complex CAD assembly, a robot arm. It includes multiple parts, two closed loops, multiple constraints, and a subassembly. Look for the 11 CAD files of this case study in the SimMechanics Link examples folder.

File Name	CAD File Type
robot.ASSEMBLYFILETYPE	Assembly
grip.ASSEMBLYFILETYPE	Subassembly (flexible)
base.PARTFILETYPE forearm.PARTFILETYPE upperarm.PARTFILETYPE wrist.PARTFILETYPE	Parts (main assembly)
fingertips.PARTFILETYPE (twice) firstfingerlink.PARTFILETYPE firstfingerlinkL.PARTFILETYPE metacarpal.PARTFILETYPE secondfingerlink.PARTFILETYPE (twice)	Parts (subassembly)

CAD Robot Arm Properties

Open the assembly file for the whole robot.



Robot Arm Assembly in a CAD Platform

In the assembly tree to the left of the window, examine the CAD hierarchy:

- Five of the part files are grouped into the subassembly `grip`. The subassembly uses two instances each of `fingertips` and `secondfingerlink`.
- The subassembly has its own group of 18 constraints, `MateGroup1`.
Two constraints, `Angle1` and `Angle2`, are not active. If they were, they would lock the grip fingers into the open position. Here, each grip finger can move separately.
- The other four part files are separate and grouped into the main assembly.

- The main assembly has its own MateGroup1, consisting of seven constraints.

The whole assembly has eight DoFs. The grip subassembly alone contains two, allowing each finger to open and close separately. The main assembly has six DoFs:

- The upper arm can move relative to the base by pitching, yawing, and rolling (three DoFs).
- The forearm can yaw relative to the upper arm (one DoF).
- The wrist can pitch relative to the forearm (one DoF).
- The grip can rotate about its symmetry axis (one DoF).

Export Robot Arm Assembly

Apply any changes you want to the assembly configuration or settings. If you change the assembly or any subassemblies, you need to rebuild the assembly before exporting it to XML.

Using the SimMechanics Link interface to your CAD platform, export the assembly into Physical Modeling XML. The XML file `robot.xml` appears in your working CAD folder.

Import Model

Once you have exported the CAD assembly, you can generate the corresponding SimMechanics model. Using the `mech_import` command, you import the Physical Modeling XML file that SimMechanics Link generated during export, and let SimMechanics automatically generate the model for you. Then, review the model and make any required changes to the model. For a step-by-step description of the CAD Import procedure, see “Import Robot Arm Model”.

Export a CAD Stewart Platform

In this section...

“About the Stewart Platform” on page 3-43

“CAD Assembly” on page 3-43

“Open Assembly” on page 3-44

“Export Assembly” on page 3-45

“Import Model” on page 3-45

About the Stewart Platform

The Stewart platform consists of two plates connected by six mobile and extensible legs. The lower or base plate is immobile. The upper or mobile plate has six degrees of freedom, three rotational and three translational. The platform is a six-degree-of-freedom (DoF) mechanical system used for accurate positioning applications. It is highly stable and easy to control.

The platform’s six legs each have two parts, an upper and a lower leg, with a piston-like cylindrical DoF between each pair of parts. The legs are connected to the base plate and the top plate by universal joints at each end of each leg. (These universals are not just sets of abstract DoFs. Each also contains a spider-like body, while also having two DoFs.) The upper part of each leg can slide into and out of the lower leg, allowing each leg to be varied in length. The position and orientation of the mobile platform (top plate) varies depending on the lengths to which the six legs are separately adjusted.

Once the top is connected to the legs, the entire Stewart platform assembly has 36 DoFs. Only six DoFs are *independent*, the same as the top plate would have if it were disconnected. You can think of these independent DoFs as the six adjustable leg lengths or as equivalent to the six DoFs of the mobile plate.

CAD Assembly

The following example uses a complex computer-aided design (CAD) assembly that models the Stewart platform.

Note The Stewart platform assembly in this example is an advanced example of computer-aided design. You should work through the previous case studies before attempting to work with this assembly.

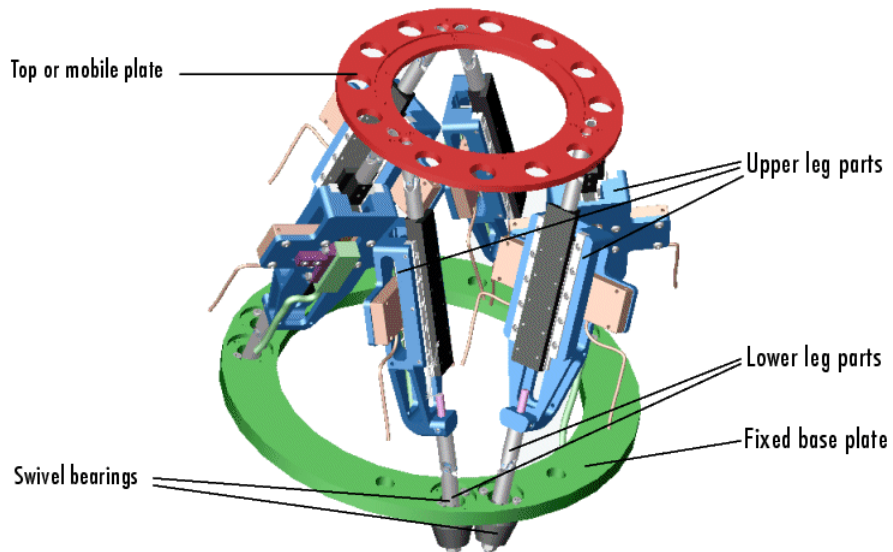
Stewart Platform Assembly Files

Look for the 45 CAD files of this case study in the *smlink/smlinkdemos* directory. The master assembly file is:

`stewart_platform.ASSEMBLYFILETYPE`

Open Assembly

Open the master assembly file, `stewart_platform.ASSEMBLYFILETYPE`. Click the assembly and rotate it to view the top and bottom plates and the legs.



Stewart Platform CAD Assembly

The CAD hierarchy for the Stewart platform contains assemblies for the top and base plates, as well as assemblies for the six legs. All the constraints on the assembly parts are grouped into one group, containing 30 constraints.

There are 448 component parts and 38 subassemblies, which you can open individually to examine the separate parts.

The base plate is about 24 centimeters (cm) in diameter; the top plate about 16.5 cm. When centered and oriented flat, the top plate is about 20 cm above the base. The assembly models the platform material as aluminum (about 2.7 grams per cubic cm).

Export Assembly

Apply any changes you want to the assembly configuration or settings. If you change the assembly or any subassemblies, you need to rebuild the assembly before exporting it to XML.

Using the SimMechanics Link interface to your CAD platform, export the assembly into Physical Modeling XML. Because the assembly is so complex, the export process takes longer than it does for simpler assemblies. As the export proceeds, various parts and subassemblies are highlighted. When the highlighting stops, the export is finished.

The exported model appears as `stewart_platform.xml` in your working CAD folder.

Import Model

Once you have exported the CAD assembly, you can generate the corresponding SimMechanics model. Using the `mech_import` command, you import the Physical Modeling XML file that SimMechanics Link generated during export, and let SimMechanics automatically generate the model for you. Then, review the model and make any required changes to the model. For a step-by-step description of the CAD Import procedure, see “Import a CAD Stewart Platform Model”.

Second-Generation Examples

- “Export Robot Assembly from SolidWorks Software” on page 4-2
- “Export Robot Assembly from Autodesk Inventor Software” on page 4-8
- “Export Robot Assembly from Creo Software” on page 4-13
- “Export Stewart Platform from SolidWorks Software” on page 4-20
- “Export Stewart Platform from Creo Software” on page 4-26

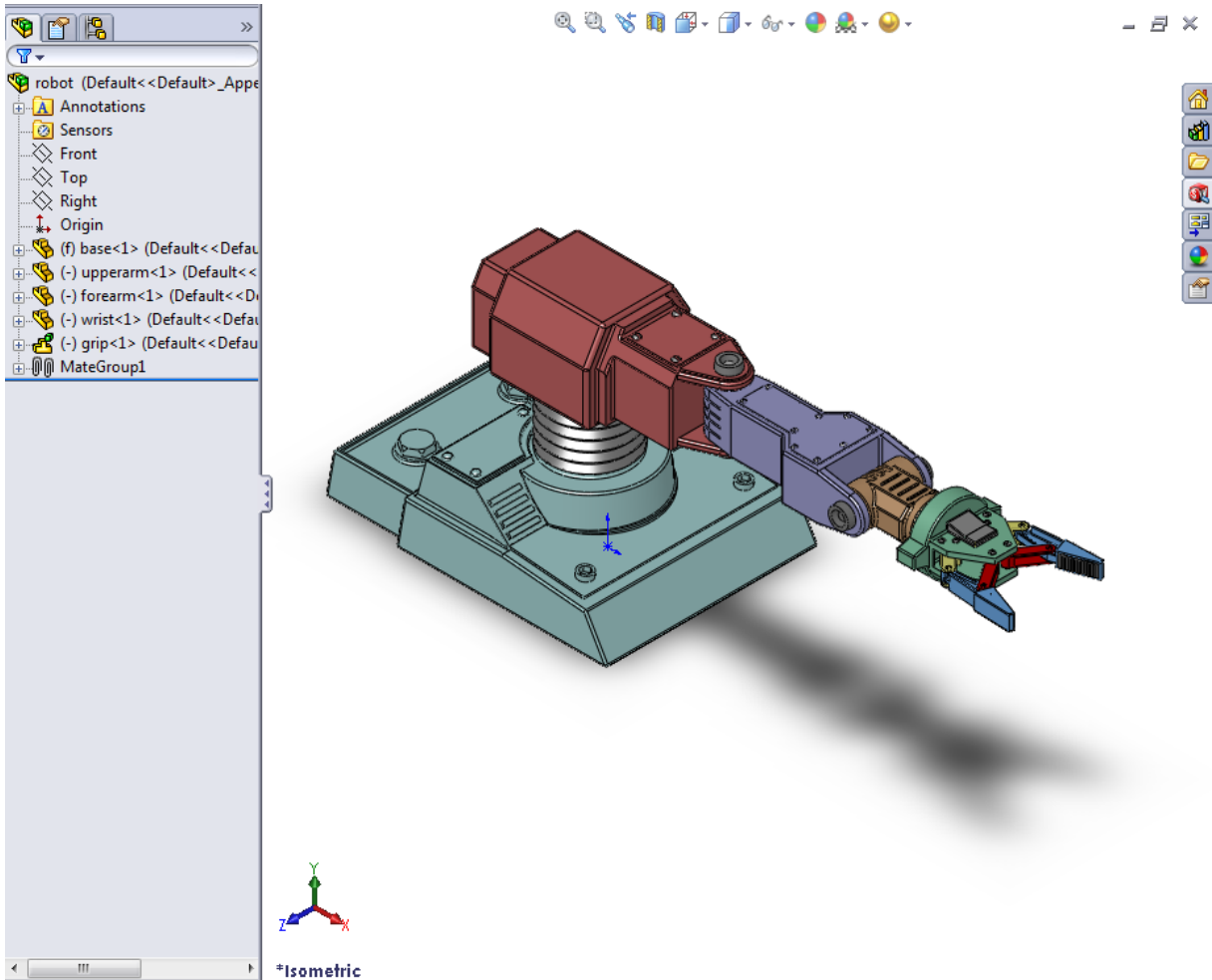
Export Robot Assembly from SolidWorks Software

In this section...
“Example Requirements” on page 4-3
“Open Robot Assembly” on page 4-4
“Export CAD Assembly” on page 4-4
“Verify CAD Export Files” on page 4-5
“About the Example CAD Files” on page 4-5
“About CAD Export” on page 4-6
“Export in First- and Second-Generation Formats” on page 4-7

In this example, you export a SolidWorks CAD assembly that represents a robot arm. The export procedure generates one XML file and a set of STL files that you can import into SimMechanics to generate a new second-generation model.

The example begins with a procedure to export the CAD assembly. Information on the robot CAD files and CAD Export follows the export procedure. To import the robot assembly into a SimMechanics Second Generation model, see “Import Robot Arm Model”.

The following figure shows the robot CAD assembly you export in this example.



Example Requirements

To successfully complete this example, you must meet the following requirements:

- Have the latest version of SimMechanics Link installed on your machine.

- Have completed the linking procedure for your SolidWorks installation. The linking procedure adds a SimMechanics Link add-in to the SolidWorks installation.
- Have access to the robot example files that accompany the SimMechanics Link installation. See “Export Robot Assembly from SolidWorks Software” on page 4-2.

Open Robot Assembly

Before you can export the robot assembly, you must load the assembly into SolidWorks.

- 1 Open SolidWorks on your machine.
- 2 Select **File > Open**
- 3 Navigate to the file directory that contains the robot CAD files.

Note The directory that contains the robot CAD files for the SolidWorks platform is

```
<MATLAB Root>\toolbox\physmod\smlink\smlinkdemos\...  
...solidworks\robot
```

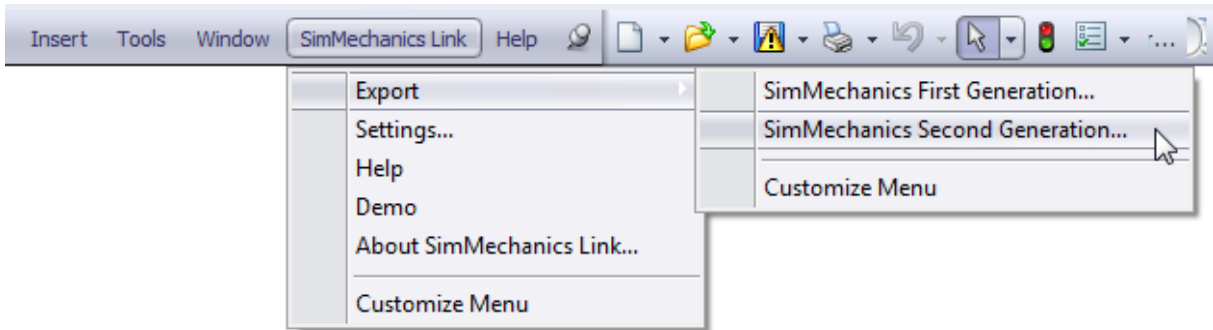
- 4 Select file robot.SLDASM.
- 5 Click **Open**.

The CAD platform opens the robot assembly.

Export CAD Assembly

Once you successfully open the robot CAD assembly in your SolidWorks installation, you can export the assembly in second-generation format:

- 1 On the SolidWorks tool bar, select **SimMechanics Link > Export** and click **SimMechanics Second Generation...**



- 2 In the **File name:** field of the **Save As** dialog box, enter `sm_robot` and select a directory to export the files.
- 3 Click **Save**.

Note Large CAD assemblies require more time to complete the export process. Allow up to a few minutes for the export process to complete.

Verify CAD Export Files

Confirm the following files exist in the specified export directory:

- XML file — provides structure of model and parameters of parts.
- STL files — provides surface geometry of parts.

About the Example CAD Files

The CAD assembly files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
<MATLAB Root>\matlab\toolbox\physmod\smlink\smlinkdemos\...
...solidworks\robot
```

Substitute <MATLAB Root> with the root directory of your MATLAB installation.

Note If you are not sure what your MATLAB root directory is, at the MATLAB command line enter `matlabroot`. MATLAB returns the root directory for your installation.

The `\robot` directory contains a set of CAD files that define each CAD part and CAD assembly. Part file names contain the file extension `.SLDPRT`. Assembly file names contain the extension `.SLDASM`. A drawing file with extension `.SLDDRW` contains robot arm dimensions.

The robot assembly contains nine parts and two assemblies: `robot.SLDASM` and `grip.SLDASM`. File `robot.SLDASM` models the robot root assembly. File `grip.SLDASM` models the robot grip subassembly.

The following table lists all files the example requires.

File Name	Type
<code>robot.SLDASM</code>	Assembly
<code>grip.SLDASM</code>	
<code>wrist.SLDPR</code>	Part
<code>upperarm.SLDPRT</code>	
<code>secondfingerlink.SLDPRT</code>	
<code>metacarples.SLDPRT</code>	
<code>firstfingerlinkL.SLDPRT</code>	
<code>forearm.SLDPRT</code>	
<code>firstfingerlink.SLDPRT</code>	
<code>fingertips.SLDPRT</code>	
<code>base.SLDPRT</code>	

About CAD Export

The CAD export procedure generates one SimMechanics Import XML file and a set of STL files. The XML file contains the structure of the assembly and

the parameters that define each part. The STL files define the 3-D geometry of each part. Once the export procedure is complete, you can import the SimMechanics Import XML file into SimMechanics. SimMechanics uses the file to automatically generate a new SimMechanics Second Generation model.

Export in First- and Second-Generation Formats

You can export a CAD assembly in first- or second-generation formats. Export in first-generation format generates a Physical Modeling XML file. Export in second-generation format generates a SimMechanics Import XML file.

When you import the CAD assembly, you must use the appropriate command for the format selected. If you export a model in first-generation format, you must use the `mech_import` command. If you export a model in second-generation format, you must use the `smimport` command.

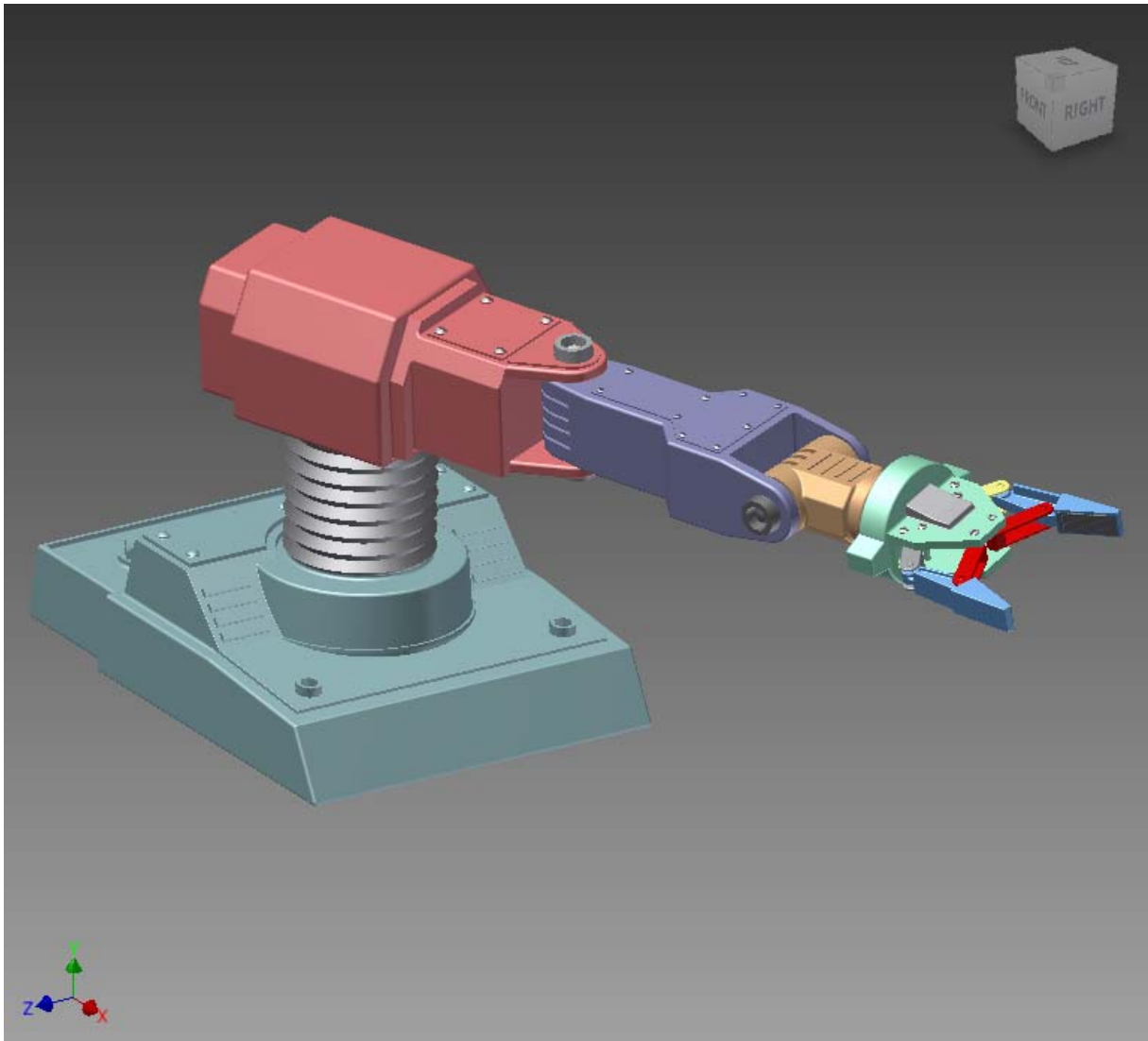
Export Robot Assembly from Autodesk Inventor Software

In this section...
“Example Requirements” on page 4-9
“Open Robot Assembly” on page 4-10
“Export CAD Assembly” on page 4-10
“Check CAD Export Files” on page 4-11
“About the Example CAD Files” on page 4-11
“About CAD Export” on page 4-12

In this example, you export an Autodesk Inventor CAD assembly that represents a robot arm. The export procedure generates one XML file and a set of STL files that you can import into SimMechanics to generate a new second-generation model.

The example begins with a procedure to export the CAD assembly. Information on the robot CAD files and CAD Export follows the export procedure. To import the robot assembly into a SimMechanics Second Generation model, see “Import Robot Arm Model”.

The following figure shows the robot CAD assembly you export in this example.



Example Requirements

To successfully complete this example, you must have:

- Installed the latest SimMechanics Link version on your machine.
- Completed the linking procedure for your Inventor installation. The linking procedure adds a SimMechanics Link add-in to the Inventor installation.

Open Robot Assembly

Before you can export the robot assembly, you must load the assembly into Inventor.

- 1** Open Inventor on your machine.
- 2** Select **File > Open**
- 3** Navigate to the file directory that contains the robot CAD files.

Note The directory that contains the robot CAD files for the Autodesk Inventor platform is

```
<MATLAB Root>\toolbox\physmod\smlink\smlinkdemos\...  
...inventor\robot
```

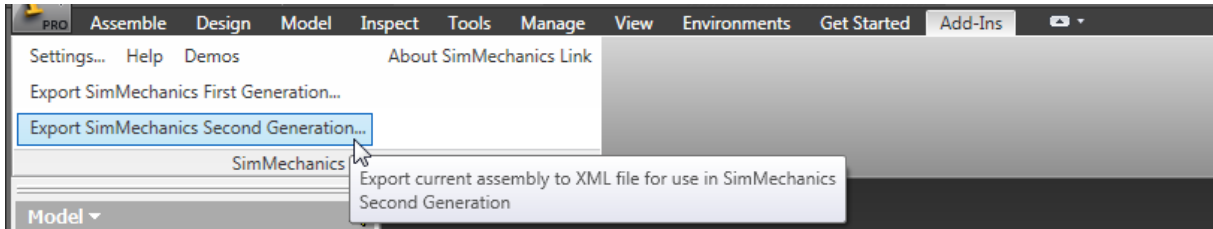
- 4** Select file `robot.IAM`.
- 5** Click **Open**.

The CAD platform opens the robot assembly.

Export CAD Assembly

Once you successfully open the robot CAD assembly in your Inventor installation, you can export the assembly in second-generation format:

- 1** In the Inventor toolbar, select **Add-Ins > Export SimMechanics Second Generation**.



- 2** In the **File name:** field of the **Save As** dialog box, enter `sm_robot` and select a directory to export the files.
- 3** Click **Save**.

Note Large CAD assemblies require more time to complete the export process. Allow up to a few minutes for the export process to complete.

Check CAD Export Files

Confirm the following files exist in the export directory you specified:

- XML file—Provides structure of model and parameters of parts.
- STL files—Provide surface geometry of parts.

About the Example CAD Files

The CAD assembly files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
<MATLAB Root>\matlab\toolbox\physmod\smlink\smlinkdemos\...
...inventor\robot
```

Substitute <MATLAB Root> with the root directory of your MATLAB installation.

Note If you are not sure what your MATLAB root directory is, at the MATLAB command line enter `matlabroot`. MATLAB returns the root directory for your installation.

The \inventor\robot directory contains a set of CAD files that define each CAD part and CAD assembly. Part file names contain the file extension .IPT. Assembly file names contain the extension .IAM.

The robot assembly contains nine parts and two assemblies: robot.IAM and grip.IAM. File robot.IAM models the robot root assembly. File grip.IAM models the robot grip subassembly.

The following table lists all files the example requires.

File Name	Type
robot.IAM	Assembly
grip.IAM	
wrist.IPT	Part
upperarm.IPT	
secondfingerlink.IPT	
metacarples.IPT	
firstfingerlinkL.IPT	
forearm.IPT	
firstfingerlink.IPT	
fingertips.IPT	
base.IPT	

About CAD Export

The CAD export procedure generates one SimMechanics Import XML file and a set of STL files. The XML file contains the structure of the assembly and the parameters that define each part. The STL files define the 3-D geometry of each part. Once the export procedure is complete, you can import the SimMechanics Import XML file into SimMechanics. SimMechanics uses the file to automatically generate a new SimMechanics Second Generation model.

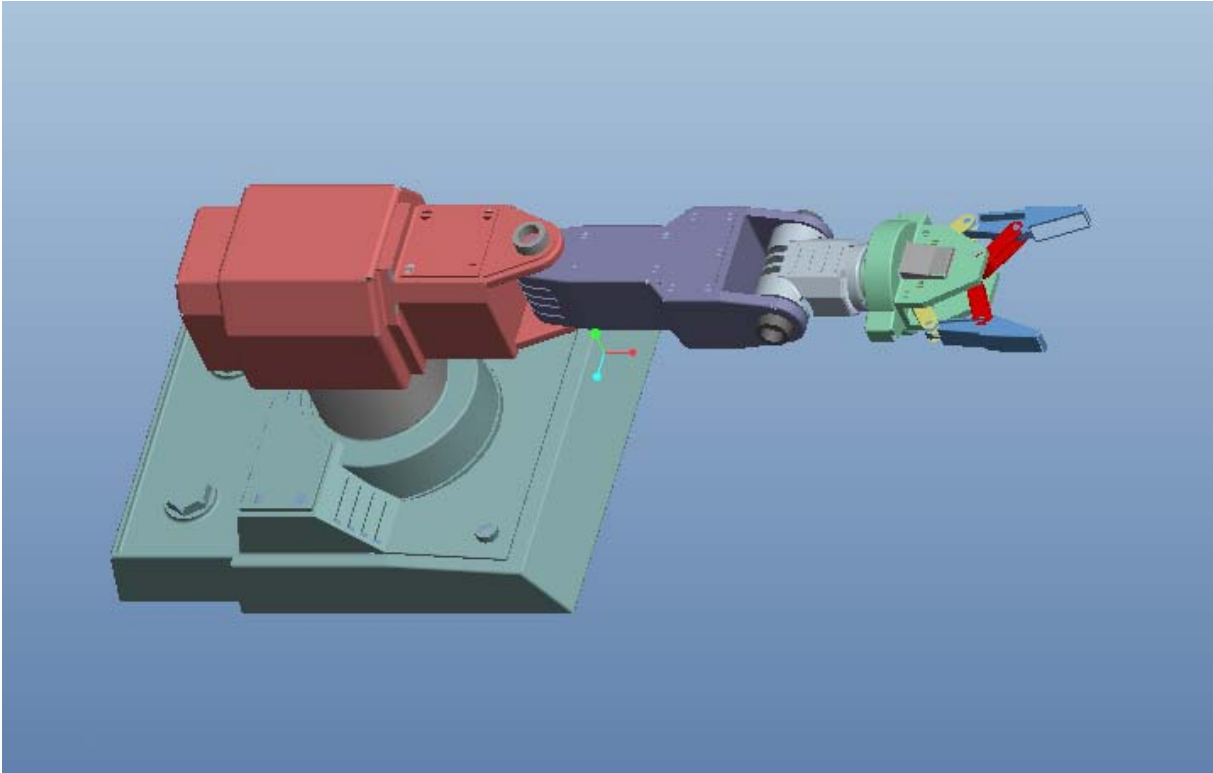
Export Robot Assembly from Creo Software

In this section...
“Example Requirements” on page 4-14
“Open Robot Assembly” on page 4-15
“Export CAD Assembly” on page 4-15
“Verify CAD Export Files” on page 4-16
“About the Example CAD Files” on page 4-17
“About CAD Export” on page 4-18
“Export in First- and Second-Generation Formats” on page 4-19

In this example, you export a Creo (Pro/ENGINEER) CAD assembly that represents a robot arm. The export procedure generates one XML file and a set of STL files that you can import into SimMechanics to generate a new second-generation model.

The example begins with a procedure to export the CAD assembly. Information on the robot CAD files and CAD Export follows the export procedure. To import the robot assembly into a SimMechanics Second Generation model, see “Import Robot Arm Model”.

The following figure shows the robot CAD assembly that you export in this example.



Example Requirements

To successfully complete this example, you must meet the following requirements:

- Have the latest version of SimMechanics Link installed on your machine.
- Have completed the linking procedure for your Creo installation. The linking procedure adds a SimMechanics Link add-on tool to the installation.
- Have access to the robot example files that accompany the SimMechanics Link installation.

Open Robot Assembly

Before you can export the robot assembly, you must load the assembly into Creo.

- 1 Open Creo on your machine.
- 2 Select **File > Open**
- 3 Navigate to the file directory that contains the robot CAD files.

Note The directory that contains the robot CAD files for the Pro/ENGINEER platform is

<MATLAB Root>\toolbox\physmod\smlink\smlinkdemos\proe\robot

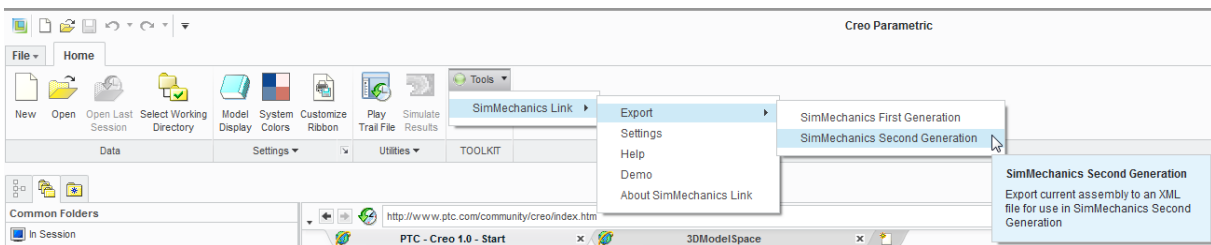
- 4 Select file robot.ASM.
- 5 Click **Open**.

The CAD platform opens the robot assembly.

Export CAD Assembly

Once you successfully open the robot CAD assembly in your Creo installation, you can export the assembly in second-generation format:

- 1 In the Creo toolbar, click **Tools**.



- 2 Select **SimMechanics Link > Export**.

3 Select the SimMechanics generation you wish to export the assembly into.

Options include **SimMechanics First Generation** and **SimMechanics Second Generation**.

4 In the **File name:** field of the **Select Folder** dialog box, enter select a directory to export the files into.

Note Do not enter a name in the **File name** field. The field should contain only a dot .

5 Click **Open**.

6 In the new dialog box, enter file name `sm_robot` and click the green arrow.

Note SimMechanics Link generates a new SimMechanics Import XML file and a set of STL files that you can import into a second-generation model.

Verify CAD Export Files

Confirm the following files exist in the export directory you specified:

SimMechanics Import XML File

Name	Type	Description
sm_robot.xml	SimMechanics Import XML File	The XML file defines the structure of the CAD assembly. The file also contains the parameters required to model each part in the CAD assembly.

STL Geometry Files

Note In the **Name** column, <identifier> is a set of numeric and alphabetic characters that identifies each file.

Name	Type	Description
base_prt.STL	STL geometry files	The STL files define the surface geometry of each robot CAD part
fingertips_prt.STL		
firstfingerlink_prt.STL		
firstfingerlinkl_prt.STL		
forearm_prt.STL		
metacarples_prt.STL		
secondfingerlink_prt.STL		
upperarm_prt.STL		
wrist_prt.STL		

About the Example CAD Files

The CAD assembly files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
<MATLAB Root>\matlab\toolbox\physmod\smlink\smlinkdemos\...
...proe\robot
```

Substitute <MATLAB Root> with the root directory of your MATLAB installation.

Note If you are not sure what your MATLAB root directory is, at the MATLAB command line enter `matlabroot`. MATLAB returns the root directory for your installation.

The \proe\robot directory contains a set of CAD files that define each CAD part and CAD assembly. Part file names contain the file extension .PRT. Assembly file names contain the extension .ASM.

The robot assembly contains nine parts and two assemblies: robot.ASM and grip.ASM. File robot.ASM models the robot root assembly. File grip.ASM models the robot grip subassembly.

The following table lists all files the example requires.

File Name	Type
robot.ASM	Assembly
grip.ASM	
wrist.PRT	Part
upperarm.PRT	
secondfingerlink.PRT	
metacarples.PRT	
firstfingerlinkL.PRT	
forearm.PRT	
firstfingerlink.PRT	
fingertips.PRT	
base.PRT	

About CAD Export

The CAD export procedure generates one SimMechanics Import XML file and a set of STL files. The XML file contains the structure of the assembly and the parameters that define each part. The STL files define the 3-D geometry of each part. Once the export procedure is complete, you can import the SimMechanics Import XML file into SimMechanics. SimMechanics uses the file to automatically generate a new SimMechanics Second Generation model.

Export in First- and Second-Generation Formats

You can export a CAD assembly in first- or second-generation formats. Export in first-generation format generates a Physical Modeling XML file. Export in second-generation format generates a SimMechanics Import XML file.

When you import the CAD assembly, you must use the appropriate command for the format selected. If you export a model in first-generation format, you must use the `mech_import` command. If you export a model in second-generation format, you must use the `smimport` command.

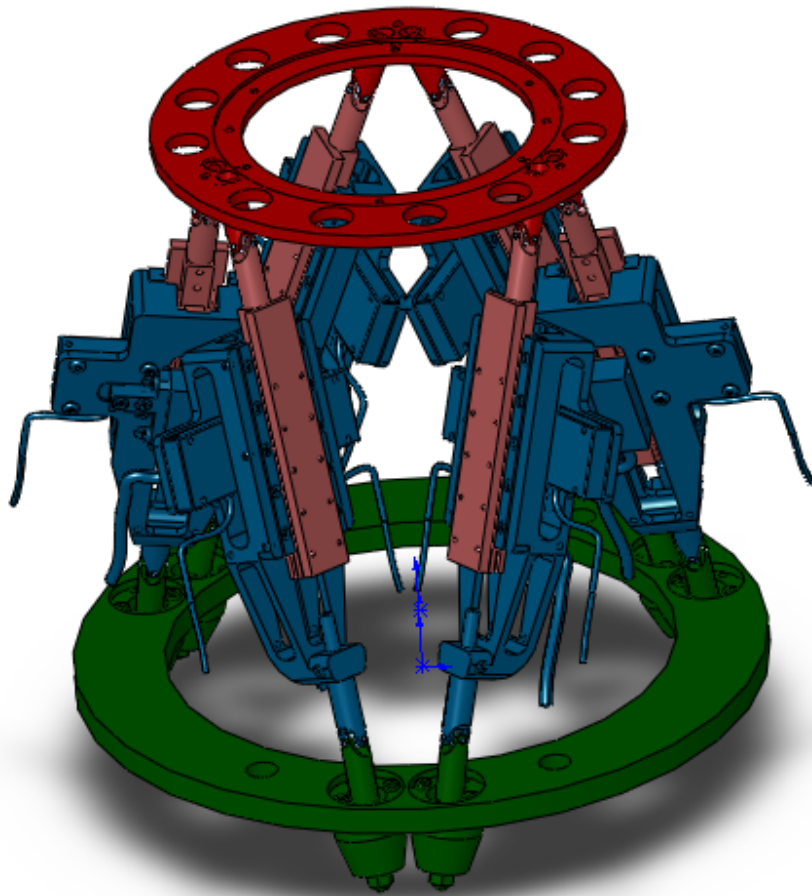
Export Stewart Platform from SolidWorks Software

In this section...
“Example Requirements” on page 4-21
“Open Robot Assembly” on page 4-22
“Export CAD Assembly” on page 4-22
“Verify CAD Export Files” on page 4-23
“About the Example CAD Files” on page 4-25
“About CAD Export” on page 4-25

In this example, you export a SolidWorks CAD assembly that represents a Stewart platform. The export procedure generates one XML file and a set of STL files that you can import into SimMechanics to generate a new second-generation model.

The example begins with a procedure to export the CAD assembly. Information on CAD files and CAD Export follows the export procedure. To import the robot assembly into a SimMechanics Second Generation model, see “Import Robot Arm Model”.

The following figure shows the Stewart platform assembly you export in this example.



Example Requirements

To successfully complete this example, you must meet the following requirements:

- Have the latest version of SimMechanics Link installed on your machine.

- Have completed the linking procedure for your SolidWorks installation. The linking procedure adds a SimMechanics Link add-in to the SolidWorks installation.
- Have access to the Stewart Platform example files that accompany the SimMechanics Link installation. See “About the Example CAD Files” on page 4-25.

Open Robot Assembly

Before you can export the robot assembly, you must load the assembly in a SolidWorks session.

- 1 Open SolidWorks on your machine.
- 2 Select **File > Open**
- 3 Navigate to the file directory that contains the robot CAD files.

Note The directory that contains the robot CAD files for the SolidWorks platform is

```
<MATLAB Root>\toolbox\physmod\smlink\smlinkdemos\...  
...solidworks\stewart
```

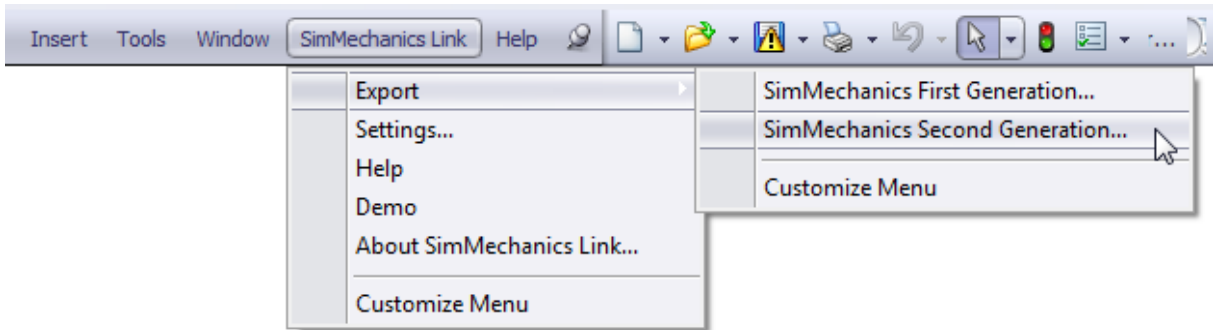
- 4 Select file `stewart_platform.SLDASM`.
- 5 Click **Open**.

The CAD platform opens the Stewart platform assembly.

Export CAD Assembly

Once you successfully open the CAD assembly in your SolidWorks installation, you can export the assembly in second-generation format:

- 1 In the SolidWorks toolbar, select **SimMechanics Link > Export** and click **SimMechanics Second Generation...**



- 2 In the **File name:** field of the **Save As** dialog box, enter `sm_stewart_platform` and select a directory to export the files.
- 3 Click **Save**.

Note Large CAD assemblies require more time to complete the export process. Allow up to a few minutes for the export process to complete.

You can proceed to import the robot assembly into a SimMechanics Second Generation model. See “Import Robot Arm Model”.

Verify CAD Export Files

Confirm the following files exist in the specified export directory:

SimMechanics Import XML File

Name	Type	Description
<code>sm_stewart_platform.xml</code>	SimMechanics Import XML File	Provides the assembly structure and part parameters that define the CAD assembly

STL Geometry Files

Name	Type	Description
BaseRing_Default_sldprt.STL	STL geometry files	Define the surface geometry of a 3-D CAD part
Bearing_Block_Default_sldprt.STL		
Carrier_Block_Default_sldprt.STL		
CenterBlock_Default_sldprt.STL		
CeramicStrip5010_Default_sldprt.STL		
Clevis_pin_A_3x6_Default_sldprt.STL		
Cylinder_Head_Cap_Screw_M3x10_Default_sldprt.STL		
Cylinder_Head_Cap_Screw_M3x8_Default_sldprt.STL		
DialPlate2_Default_sldprt.STL		
GratingStrip5010_Default_sldprt.STL		
HR_1_Motor_Default_sldprt.STL		
Hex_Nut_M5_Default_sldprt.STL		
Hexagon_Socket_Button_Head_Screw_M3x6_Default_sldprt.STL		
Hexagon_Socket_Head_Cap_Screw_M3x5_Default_sldprt.STL		
Hexagon_Socket_Set_Screw_M3x3_Default_sldprt.STL		
MPB_C345MB_Default_sldprt.STL		
RGH25_Default_sldprt.STL		
Recessed_Pan_Head_Screw_M2x8_48_Z_Default_sldprt.STL		
RenishawMount_Default_sldprt.STL		
SV2075_Default_sldprt.STL		
ShaftStop_Default_sldprt.STL		
Spider_Default_sldprt.STL		
Threaded_Shaft_Default_sldprt.STL		
Yoke_Default_sldprt.STL		
pin2_Default_sldprt.STL		

Name	Type	Description
stewart_platform.xml		

About the Example CAD Files

The CAD assembly files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
<MATLAB Root>\matlab\toolbox\physmod\smlink\smlinkdemos\...
...solidworks\stewart
```

Substitute <MATLAB Root> with the root directory of your MATLAB installation.

Note If you are not sure what your MATLAB root directory is, at the MATLAB command line enter `matlabroot`. MATLAB returns the root directory for your installation.

The `\stewart` directory contains a set of CAD files that define each CAD part and CAD assembly. Part file names contain the file extension `.SLDPRT`. Assembly file names contain the extension `.SLDASM`.

About CAD Export

The CAD export procedure generates one SimMechanics Import XML file and a set of STL files. The XML file contains the structure of the assembly and the parameters that define each part. The STL files define the 3-D geometry of each part. Once the export procedure is complete, you can import the SimMechanics Import XML file into SimMechanics. SimMechanics uses the file to automatically generate a new SimMechanics Second Generation model.

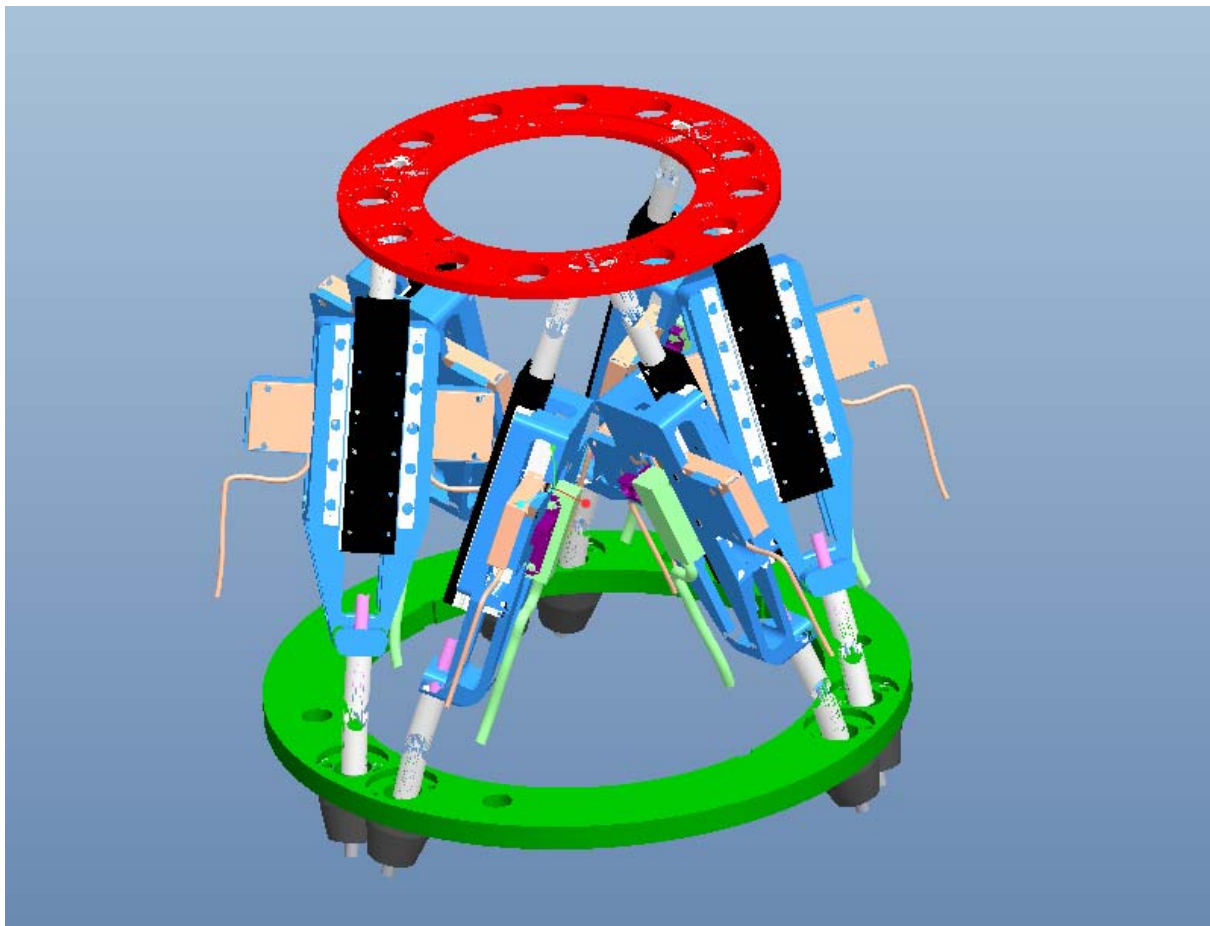
Export Stewart Platform from Creo Software

In this section...
“Example Requirements” on page 4-27
“Open Robot Assembly” on page 4-28
“Export CAD Assembly” on page 4-28
“Verify CAD Export Files” on page 4-29
“About the Example CAD Files” on page 4-31
“About CAD Export” on page 4-31
“Export in First- and Second-Generation Formats” on page 4-31

In this example, you export a Stewart platform assembly from a Creo (Pro/ENGINEER) CAD platform. The export procedure generates one XML file and a set of STL files that you can import into SimMechanics to generate a new second-generation model.

The example begins with a procedure to export the CAD assembly. Information on CAD files and CAD Export follows the export procedure. To import the robot assembly into a SimMechanics Second Generation model, see “Import Robot Arm Model”.

The following figure shows the Stewart platform assembly you export in this example.



Example Requirements

To successfully complete this example, you must meet the following requirements:

- Have the latest version of SimMechanics Link installed on your machine.
- Have completed the linking procedure for your Creo installation. The linking procedure adds a SimMechanics Link add-in to the Creo installation.

- Have access to the Stewart Platform example files that accompany the SimMechanics Link installation. See “About the Example CAD Files” on page 4-31.

Open Robot Assembly

Before you can export the robot assembly, you must load the assembly in a Creo session.

- 1 Open Creo on your machine.
- 2 Select **File > Open**
- 3 Navigate to the file directory that contains the robot CAD files.

Note The directory that contains the robot CAD files for the Creo platform is

<MATLAB Root>\toolbox\physmod\smlink\smlinkdemos\proe\stewart

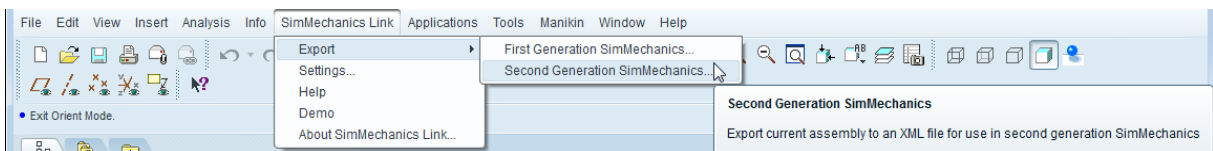
- 4 Select file `stewart_platform.ASM`.
- 5 Click **Open**.

The CAD platform opens the Stewart platform assembly.

Export CAD Assembly

Once you successfully open the CAD assembly in your Creo installation, you can export the assembly in second-generation format:

- 1 In the Creo toolbar, select **SimMechanics Link > Export** and click **SimMechanics Second Generation...**



- 2** In the **File name:** field of the **Select Folder** dialog box, enter select a directory to export the files into.

Note Do not enter a name in the **File name** field. The field should contain only a dot .

- 3** Click **Open**.

- 4** In the new dialog box, enter file name `sm_stewart_platform` and click the green arrow.

Note SimMechanics Link generates a new SimMechanics Import XML file and a set of STL files that you can import into a second-generation model.

You can proceed to import the robot assembly into a SimMechanics Second Generation model. The import model assumes the SimMechanics Import XML file you generated in this example has the name `sm_stewart_platform.xml`. If the file has a different name, change the name to `sm_stewart_platform` before proceeding. See “Import Robot Arm Model”.

Verify CAD Export Files

Confirm the following files exist in the specified export directory:

SimMechanics Import XML File

Name	Type	Description
<code>sm_stewart_platform.xml</code>	SimMechanics Import XML File	Provides the assembly structure and part parameters that define the CAD assembly

STL Geometry Files

Name	Type	Description
ai_f100128_prt.STL	STL geometry files	Define the surface geometry of a 3-D CAD part
basing_prt.STL		
bearing_block_prt.STL		
carrier_block_prt.STL		
centerblock_prt.STL		
ceramicstrip5010_prt.STL		
clevis_pin_a_3x6_prt.STL		
cylinder_head_cap_screw_m3x8_prt.STL		
cylinder_head_cap_screw_m3x10_prt.STL		
dialplate2_prt.STL		
gratingstrip5010_prt.STL		
hex_nut_m5_prt.STL		
hexagon_socket_button_head_scre_prt.STL		
hexagon_socket_head_cap_screw_m_prt.STL		
hexagon_socket_set_screw_m3x3_prt.STL		
hr_1_motor_prt.STL		
mpb_c345mb_prt.STL		
pin2_prt.STL		
recessed_pan_head_screw_m2x8_48_prt.STL		
renishawmount_prt.STL		
rgh25_prt.STL		
shaftstop_prt.STL		
spider_prt.STL		
sv2075_prt.STL		
threaded_shaft_prt.STL		

Name	Type	Description
yoke_prt.STL		

About the Example CAD Files

The CAD assembly files are present in your SimMechanics Link installation. You can access the files in the following directory:

```
<matlabroot>\matlab\toolbox\physmod\smlink\smlinkdemos\...
...proe\stewart
```

Substitute <matlabroot> with the root directory of your MATLAB installation.

Note If you are not sure what your MATLAB root directory is, at the MATLAB command line enter `matlabroot`. MATLAB returns the root directory for your installation.

The `\stewart` directory contains a set of CAD files that define each CAD part and CAD assembly. Part file names contain the file extension `.PRT`. Assembly file names contain the extension `.ASM`.

About CAD Export

The CAD export procedure generates one SimMechanics Import XML file and a set of STL files. The XML file contains the structure of the assembly and the parameters that define each part. The STL files define the 3-D geometry of each part. Once the export procedure is complete, you can import the SimMechanics Import XML file into SimMechanics. SimMechanics uses the file to automatically generate a new SimMechanics Second Generation model.

Export in First- and Second-Generation Formats

You can export a CAD assembly in first- or second-generation formats. Export in first-generation format generates a Physical Modeling XML file. Export in second-generation format generates a SimMechanics Import XML file.

When you import the CAD assembly, you must use the appropriate command for the format selected. If you export a model in first-generation format, you must use the `mech_import` command. If you export a model in second-generation format, you must use the `smimport` command.

Custom Linking to Third-Party Applications

The SimMechanics Link software includes an application programming interface (API). With the API of an external application program and the SimMechanics Link API, you can selectively transfer data from the external application to the SimMechanics Link software.

- “Custom Export with SimMechanics Link API” on page 5-2
- “Custom Translation into Physical Modeling XML Format” on page 5-6
- “Design Custom Exporter Module” on page 5-11
- “Program Custom Exporter with SimMechanics Link API” on page 5-15

Custom Export with SimMechanics Link API

In this section...

“About Custom CAD Export” on page 5-2

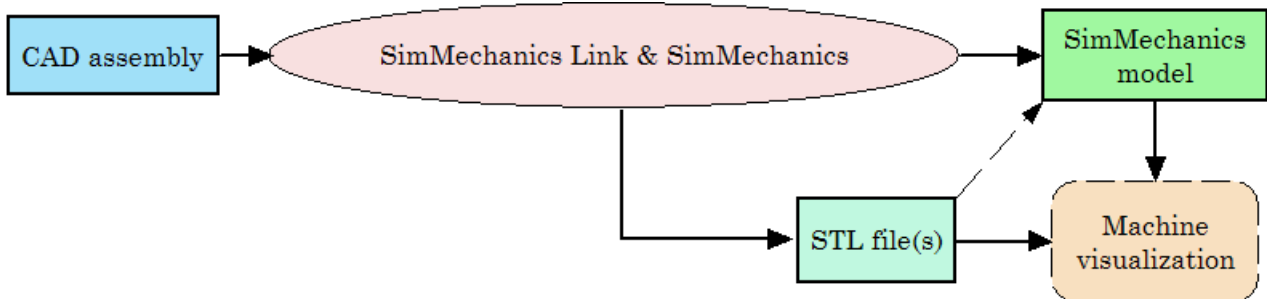
“Custom Translation Steps in Common with Standard Export” on page 5-3

“Custom Translation Steps Different from Standard Export” on page 5-4

“Requirements for Creating a Custom Exporter” on page 5-4

About Custom CAD Export

The translation process from an externally defined machine representation to a SimMechanics model is the same whether you use the *standard* export with a supported external third-party platform or create your own *custom* exporter. The SimMechanics Link exporter translates mechanical system data from an external application, such as a computer-aided design (CAD) platform. You can use this translated data to generate a SimMechanics model of your original mechanical system.



Reasons for Custom CAD Export

You must create and use your own custom exporter when:

- You want to export an assembly from a CAD platform that the SimMechanics Link utility does not support.
- The standard export from a supported CAD platform does not yield the results that you want.

Prerequisites for Custom CAD Export

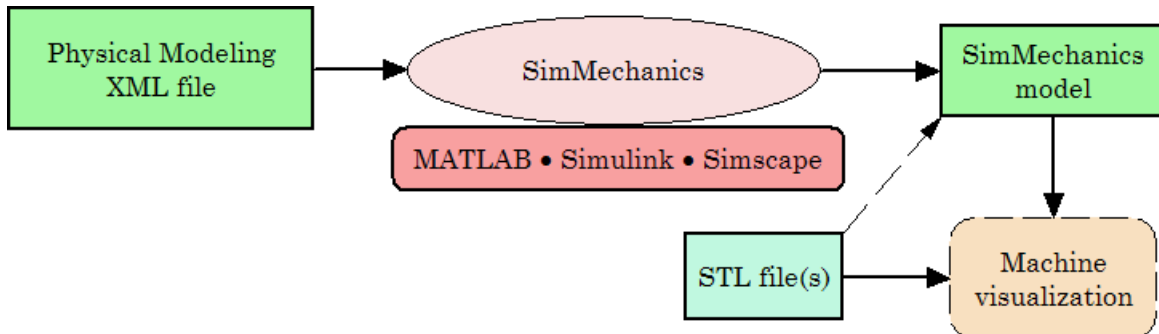
CAD translation includes CAD assembly export and SimMechanics model import. Export requires:

- A CAD platform or application, including access to its application programming interface (API)
- MATLAB installed and registered as a server
- The SimMechanics Link utility, which includes its own API
- A *custom exporter module* that you create and that transfers assembly information from the CAD platform API to the SimMechanics Link API

Import requires SimMechanics software.

Custom Translation Steps in Common with Standard Export

You generate SimMechanics models from Physical Modeling XML files. You use mechanical import and then visualize the models with stereolithographic (STL) files. *Custom* export creates an XML file to represent a CAD assembly and a set of STL files to represent the surface geometries of the assembly bodies. This part of CAD translation is the same as *standard* export.

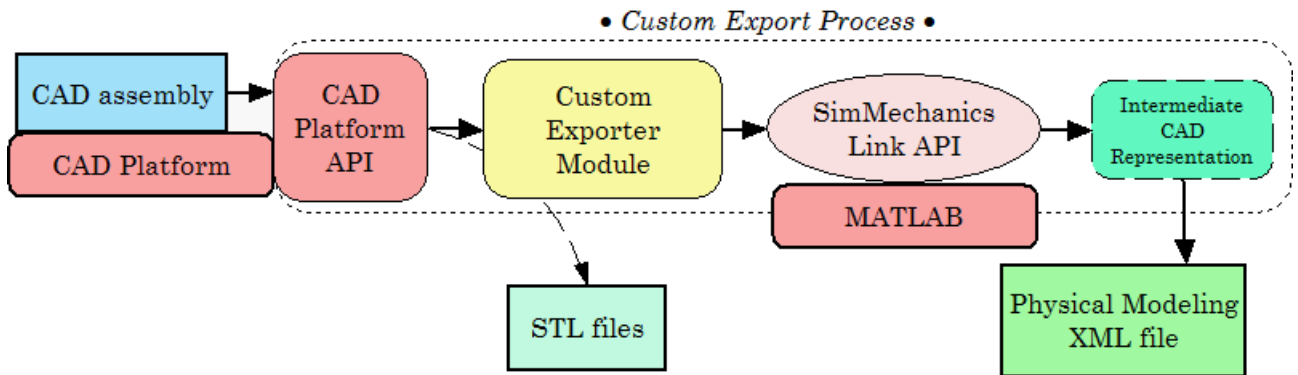


From Physical Modeling XML to Visualizable SimMechanics™ Model

Custom Translation Steps Different from Standard Export

For *custom* export, you must create a custom exporter module that interacts with the APIs of both the CAD platform and the SimMechanics Link utility. From a CAD assembly, this custom exporter module creates a selective, intermediate representation of the machine that persists for one session. The module then finishes by creating the same files as the standard export does:

- A Physical Modeling XML file representing selected data required to generate a SimMechanics model, which is written from the selective representation
- A set of STL files to represent the surface geometries of the assembly bodies



Custom Export: From CAD Assembly to Physical Modeling XML

Requirements for Creating a Custom Exporter

Note The SimMechanics Link API is supported on all platforms that MATLAB supports.

To create a custom exporter:

- Access the API of your CAD platform and understand how to call it from an external program. See your CAD platform documentation.

- Access the SimMechanics Link API and understand how to call it from an external program.
- Write the custom exporter module in C/C++.

The module might be an executable or a linked library, depending on your CAD platform requirements and API.

For more information, see “Custom Translation into Physical Modeling XML Format” on page 5-6

Custom Translation into Physical Modeling XML Format

In this section...

“About Mapping API Objects from CAD Format to Physical Modeling XML” on page 5-6

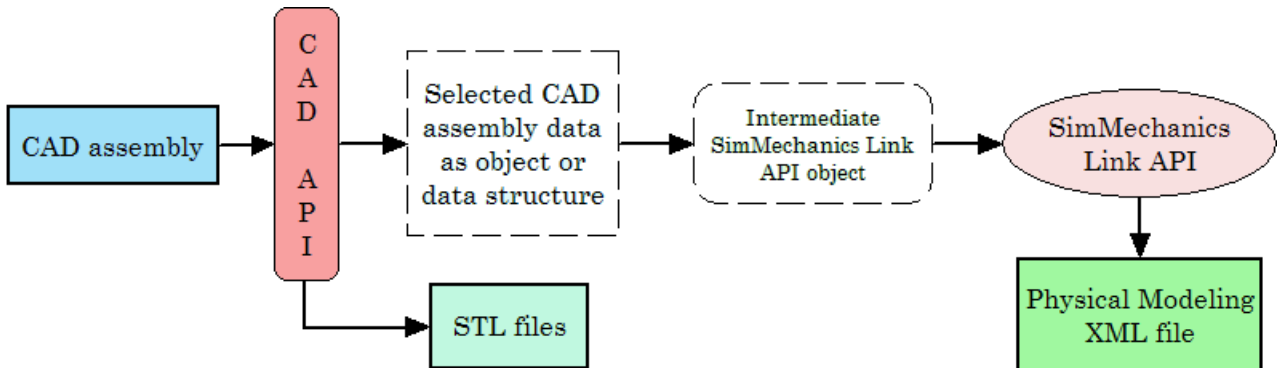
“Selecting CAD Assembly Data for Export” on page 5-8

“Constructing Intermediate API Representations” on page 5-8

“Converting Selective API Representations into Physical Modeling XML” on page 5-10

About Mapping API Objects from CAD Format to Physical Modeling XML

A complete CAD assembly contains information both relevant and not relevant for exporting a Physical Modeling XML file to represent your assembly.



For detailed information on the design of a custom export module, see section “Design Custom Exporter Module” on page 5-11.

Data Required from a CAD Assembly to Create a SimMechanics Model

You obtain the selected assembly data through the CAD platform application programming interface (API) in the form of an object or a data structure. The data required from an assembly to construct a SimMechanics model include:

Mapping of Selected Mechanical Data to SimMechanics Model Components

Data Selected from CAD Assembly or Other External Machine Definition Mapped to...	...Required SimMechanics Model Component
Assembly and parts origin	Ground and World coordinate system
Assembly and subassembly hierarchy	Model and subsystem hierarchy
Part masses and inertia tensors, colors, and body geometry file names	Bodies, including their mass properties, geometries, and colors
Constraint restrictions on relative positions and motions of parts	Joints between pairs of Bodies Body coordinate systems connected to Joints

Using the Exporter Functions to Complete the Mapping of CAD Assembly to Physical Modeling XML

Refer to the table, Mapping of Selected Mechanical Data to SimMechanics™ Model Components on page 5-7. Using your CAD exporter module, you transform the selected data listed in the first column of the table into the form required in the second column with the SimMechanics Link API functions and data types. You collect the selected data into an intermediate SimMechanics Link API object. With the SimMechanics Link API, you then write the intermediate API representation to a Physical Modeling XML file.

See “Design Custom Exporter Module” on page 5-11 following for detailed information on these API functions and data types.

Body Geometry Files Exported Directly from CAD API

For each moving Body in the model, the final SimMechanics model requires a separate body geometry file in STL format. You export these body geometry files directly from the CAD API, with file names matching the body geometry file names specified in the exported XML file. You specify these file names in the SimMechanics Link API object.

Tip You need body geometry files only for individual moving CAD parts or rigid subassemblies, not for flexible subassemblies.

Selecting CAD Assembly Data for Export

Tracing the Assembly Hierarchy: Nodes and Components

The assembly data that you need to retrieve through the CAD API starts with the assembly-subassembly hierarchy, with nodes. Each node in the hierarchical tree is an assembly *component* (a part or a subassembly).

For an individual component, the next component immediately up in the hierarchy is its *parent*. All dependent components connected below in the hierarchy are its *children*. The top parent in the assembly hierarchy is the whole assembly itself.

Identifying and Extracting Component Information

For each component, determine:

- Its geometric transformation with respect to its parent, its mass, its inertia tensor, its body geometry and color (for a part, or for a rigid subassembly exported as a single rigid body)
- The constraints that restrict how its children can move with respect to each other
- Whether or not the component is rigid or flexible; that is, whether all its children are to be treated as a single rigid body or as individual moving bodies
- Whether or not the component is *fixed*; that is, whether it is rigid or moving with respect to its parent

Constructing Intermediate API Representations

Before you can export the CAD assembly, you must construct a selective intermediate CAD representation, a collection of objects that capture the required CAD data.

Objects Required for Intermediate Representations

- *CAD models* (not to be confused with CAD assemblies or SimMechanics models). These represent assemblies, subassemblies, and parts.
- CAD model references
- Components (parts or subassemblies)
- Constraints
- A translator object constructed from all the other objects

Putting Objects Together into Intermediate Representations

To construct a full CAD representation:

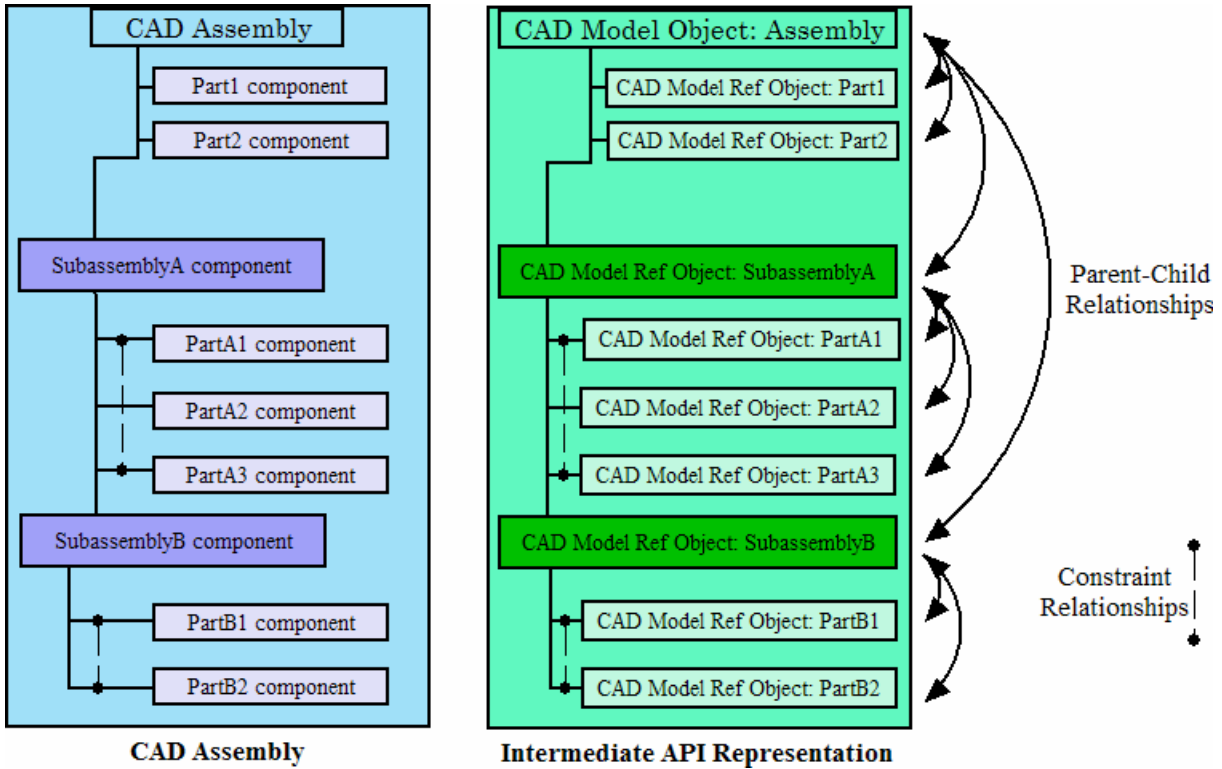
- 1** Represent the entire assembly with a CAD model object.
- 2** Represent child parts and subassemblies with additional CAD model objects.

If a subassembly is flexible (its parts can move with respect to each other), the subassembly is a parent to child parts, represented by their own CAD model objects.

- 3** Add child CAD model reference objects to their parent CAD model objects, including parent-to-child transforms.

You might need to repeat steps 1 through 3 recursively until you have mapped the entire CAD assembly hierarchy.

- 4** Represent constraints between components with constraint objects. These are also children to parent assembly and subassembly objects.
- 5** Form a single translator object from all these objects, referenced by one another in a hierarchy that parallels the original assembly hierarchy.



CAD Assembly and Intermediate API Representation: Parallel Hierarchies

Converting Selective API Representations into Physical Modeling XML

The final translator object is the last intermediate form of the CAD assembly translation. From this translator object, you create the Physical Modeling XML file representing the assembly.

Design Custom Exporter Module

In this section...

“Prerequisites for Custom Exporter Modules” on page 5-11

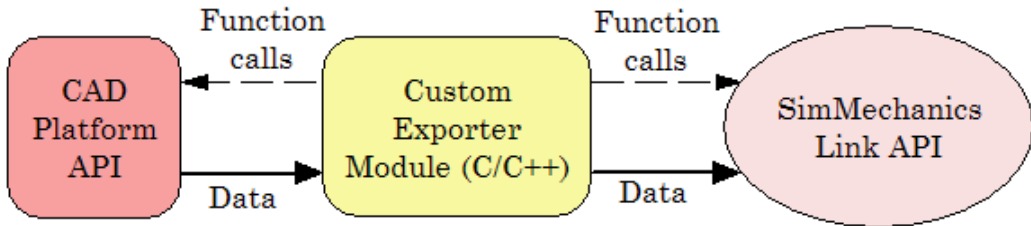
“Implementing Translation with CAD and SimMechanics Link APIs” on page 5-12

Prerequisites for Custom Exporter Modules

A custom exporter module works with intermediate representations of CAD assembly data. For an overview of the types of CAD assembly and SimMechanics Link data and their relationship, see “Custom Translation into Physical Modeling XML Format” on page 5-6.

A custom exporter module:

- Issues function calls to the APIs of both the CAD platform and the SimMechanics Link utility.
- Transfers assembly representation data from the assembly, through the CAD API, to the Physical Modeling XML file, through the SimMechanics Link API.



Custom Exporter Module Communicates with APIs (Detail)

Using the SimMechanics Link API Functions

The SimMechanics Link API is a library of API functions that you call to construct a unified, selective representation of an assembly and write it as an XML file. “Program Custom Exporter with SimMechanics Link API” on page 5-15 describes how to create, compile, and execute custom exporter modules in C/C++ with the SimMechanics Link API function library.

Implementing Translation with CAD and SimMechanics Link APIs

Extracting Selective CAD Assembly Data

To extract the selected assembly data required for export, consult your CAD platform API documentation and determine which functions of the CAD platform API you need to accomplish this.

Constructing the Intermediate API Representation

To construct the intermediate representation of an assembly, use the following functions from the SimMechanics Link API, in the order shown. The API also contains other functions you might need in special cases. For more information, consult the SimMechanics Link Reference documentation.

Step	To...	Use SimMechanics Link API Function...	...and SimMechanics Link API Objects
1.	Set length and mass units for CAD assembly. The defaults are meters (m) and kilograms (kg), respectively.	<code>pmit_set_units</code>	—
2.	a. Create CAD model object to contain data for CAD assembly	<code>pmit_create_cadmodel</code>	<code>PmitCadModelH</code>
	b. Create API representation translator object	<code>pmit_create_cad2sm</code>	<code>PmitCad2SMH</code>
3.	Set translational, rotational, and numerical tolerances for CAD assembly	<code>pmit_set_tolerances</code>	<code>PmitCad2SMH</code>

Step	To...	Use SimMechanics Link API Function...	...and SimMechanics Link API Objects
4.	a. Create object to contain data for CAD subassemblies and parts	<code>pmit_create_cadmodel</code>	<code>PmitCadModelH</code>
	b. Create reference to CAD model object	<code>pmit_create_cadmodelref</code>	<code>PmitCadModelH,</code> <code>PmitCadModelRefH</code>
	c. Add reference within existing CAD model object to child model object by adding transform	<code>pmit_add_refincadmodel</code>	<code>PmitCadModelH,</code> <code>PmitCadModelRefH</code>
5.	a. Create object to contain data for CAD assembly component (part or rigid subassembly)	<code>pmit_create_assemcomp</code>	<code>PmitAssemCompH</code>
	b. Add reference within existing CAD model object to child assembly component object	<code>pmit_add_refincomp</code>	<code>PmitAssemCompH,</code> <code>PmitCadModelRefH</code>
6.	a. Create object to contain data for constraint between CAD components (parts or rigid subassemblies)	<code>pmit_create_constrain</code>	<code>PmitAssemCompH,</code> <code>PmitConstrainH,</code> <code>PmitConstrainType,</code> <code>PmitGeomType</code>
	b. Add constraint object to CAD model object	<code>pmit_add_constrain</code>	<code>PmitCadModelH,</code> <code>PmitConstrainH</code>

Export Intermediate API Representation into Physical Modeling XML

To export the selective representation of your assembly into a Physical Modeling XML file, use the following functions from the SimMechanics Link API, in the order shown.

Step	To...	Use SimMechanics Link API Function...	...and SimMechanics Link API Objects
1.	Enable translation of CAD assembly data into SimMechanics model	<code>pmit_create_cad2sm</code>	<code>PmitCad2SMH</code>
2.	Write Physical Modeling XML file from selective CAD representation of assembly	<code>pmit_write_xml</code>	<code>PmitCad2SMH</code>

For detailed information on the programming of a custom export module with the SimMechanics Link API, see “Program Custom Exporter with SimMechanics Link API” on page 5-15

Program Custom Exporter with SimMechanics Link API

In this section...

“Including, Linking to, and Calling the API Function Library” on page 5-15

“Locating API Code Examples” on page 5-16

“A Custom Exporter Module Example” on page 5-16

Including, Linking to, and Calling the API Function Library

In the following procedures, replace:

- *matlabroot* with the root of your MATLAB installation.
- *ARCH* with the specific operating system architecture abbreviation for your system (for example, win32 for 32-bit Windows).

Including and Using the API Header File

To write your C/C++ custom exporter module, use the C header file located in *matlabroot/toolbox/physmod/smlink/api/include/*.

This header file contains the function definitions that you follow to call the SimMechanics Link API functions from your module. Include this header file in your module C/C++ source code.

Compiling and Linking the Custom Exporter Module

In addition to linking your custom module to the SimMechanics Link API, consult your CAD platform’s API documentation to determine requirements for linking to the platform’s API.

On Windows Platforms. After you compile your module, link it to the binary API function library located in *matlabroot\toolbox\physmod\smlink\api\lib*.

On UNIX, LINUX, and Mac Platforms. After you compile your module, link it to the binary API function library:

`matlabroot/bin/ARCH/libmwpmi_api.OS-SPECIFIC-EXTENSION`

`OS-SPECIFIC-EXTENSION` is the API file extension specific to your operating system.

Executing the Custom Exporter Module

To determine if you need to include your CAD platform API on the path, consult your CAD platform API documentation.

On Windows. Before you start execution of the module, verify that the folder `matlabroot\bin\ARCH\` is on the path.

On UNIX, LINUX, and Mac Platforms. Before you start execution of the module, verify that the folder `matlabroot/bin/ARCH/` is included in the system path environment variable `LD_LIBRARY_PATH`.

Locating API Code Examples

You can find API examples in this folder:

`matlabroot/toolbox/physmod/smlink/api/example/`

The subfolder, `cadapi_example/`, contains CAD-based examples.

A Custom Exporter Module Example

The folder

`matlabroot/toolbox/physmod/smlink/api/example/cadapi_example/` contains a simple C++ example of a custom module based on the API, `cadapi_example.cpp`.

- The example custom module converts two CAD parts with one constraint into two SimMechanics bodies and one joint and then writes the result to a Physical Modeling XML file.
- The example program defines the CAD assembly data internally, rather than taking the data from a CAD platform through the platform API. Compared to the figure, Custom Exporter Module Communicates with APIs (Detail) on page 5-11, the leftmost portion of the diagram is missing.

- To extend the example module to process real assembly data, add code that calls a CAD platform's API for assembly data to replace the internally defined data.

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